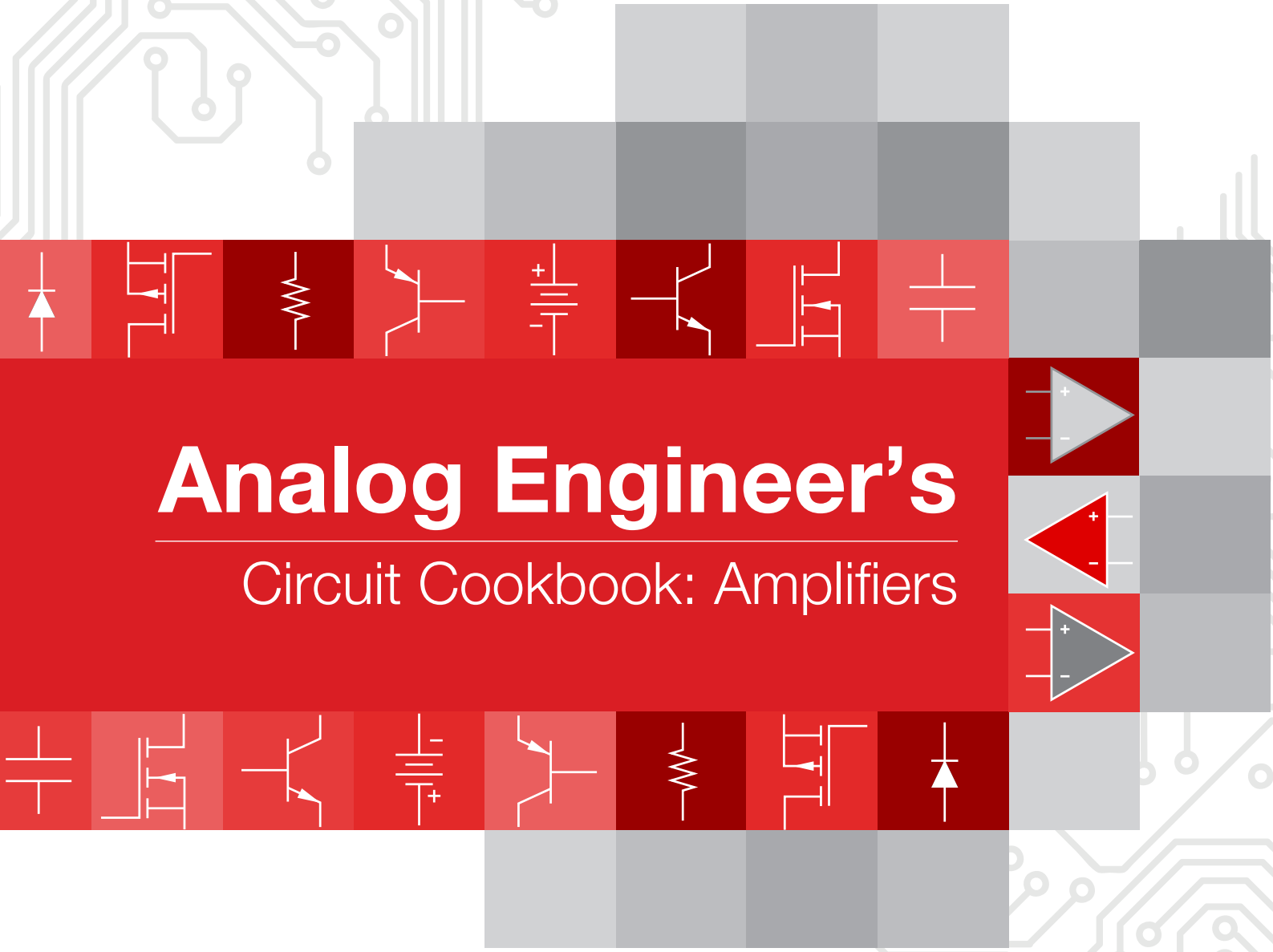


Analog Engineer's

Circuit Cookbook: Amplifiers

Texas Instruments

Analog Engineer's Circuit Cookbook: Amplifiers

Second Edition

SLYY137 - 03/2019

Edited by:

Tim Green, Pete Semig and Collin Wells

Special thanks for technical contribution:

Tim Claycomb

Mamadou Diallo

Peter Iliya

Zak Kaye

Errol Leon

Marc Liu

Masashi Miyagawa

Gustaf Falk Olson

Bala Ravi

Takahiro Saito

Will Wang

Analog Engineer's Circuit Cookbook: Amplifiers

(Second Edition)

Message from the editors:

The *Analog Engineer's Circuit Cookbook*: *Amplifiers* provides amplifier sub-circuit ideas that can be quickly adapted to meet your specific system needs. Each circuit is presented as a "definition-by-example." They include step-by-step instructions, like a recipe, with formulas enabling you to adapt the circuit to meet your design goals. Additionally, all circuits are verified with SPICE simulations.

We've provided at least one recommended amplifier for each circuit, but you can swap it with another device if you've found one that's a better fit for your design. You can search our large portfolio of amplifiers at ti.com/amplifiers.

Our circuits require a basic understanding of amplifier concepts. If you're new to amplifier design, we highly recommend completing our TI Precision Labs (TIPL) training series. TIPL includes courses on introductory topics, such as device architecture, as well as advanced, application-specific problem-solving, using both theory and practical knowledge. Check out our curriculum for operational amplifiers (op amps), analog-to-digital converters (ADCs) and more at: ti.com/precisionlabs.

We hope you find this collection of amplifier circuits helpful in developing your designs. Our goal is to regularly update the cookbook with valuable amplifier circuit building blocks. You can check to see if your version is the latest at ti.com/circuitcookbooks. If you have input on any of the existing circuits or would like to request additional amplifier cookbook circuits for the next edition please contact us at: opampcookbook@list.ti.com.

Additional resources to explore

TI Precision Labs
ti.com/precisionlabs
- On-demand courses and tutorials ranging from introductory to advanced concepts that focus on application-specific problem solving
- Hands-on labs and evaluation modules (EVM) available
  - TIPL Op Amps experimentation platform, ti.com/TIPL-amp-evm
  - TIPL SAR ADC experimentation platform, ti.com/TIPL-Data Converters-evm

Analog Engineer's Pocket Reference
ti.com/analogrefguide
- PCB, analog and mixed-signal design formulae; includes conversions, tables and equations
- e-book, iTunes and Android apps and hardcopy available

***The Signal* e-book**
ti.com/signalbook
- Op amp e-book with short, bite-sized lessons on design topics such as offset voltage, input bias current, stability, noise and more

TI Designs
ti.com/tidesigns
- Ready-to-use reference designs with theory, calculations, simulations schematics, PCB files and bench test results

TINA-TI™ Simulation Software
ti.com/tool/tina-ti
- Complete SPICE simulator for DC, AC, transient and noise analysis
- Includes schematic entry and post-processor for waveform math

Analog Engineer's Calculator
ti.com/analogcalc
- ADC and amplifier design tools, noise and stability analysis, PCB and sensor tools

Analog Wire Blog
ti.com/analogwire
- Technical blogs written by analog experts that include tips, tricks and design techniques

TI E2E™ Community
ti.com/e2e
- Support forums for all TI products

Op Amp Circuit Quick Search and Parametric Search
ti.com/opamp-search
- Search our operational amplifier portfolio by entering key parameters or by selecting a circuit function

Op Amp Parametric Cross-Reference
ti.com/opampcrossreference
- Find similar TI Amplifiers using competitive part numbers

DIY Amplifier Circuit Evaluation Module (DIYAMP-EVM)
ti.com/DIYAMP-EVM
- Single-channel circuit evaluation module providing SC70, SOT23 and SOIC package options in 12 popular amplifier configurations

Dual-Channel DIY Amplifier Circuit Evaluation Module (DUAL-DIYAMP-EVM)
ti.com/dual-diyamp-evm
- Dual-channel circuit evaluation module in an SOIC-8 package with 10 popular amplifier configurations

Want more circuits?
- **Download the *Analog Engineer's Circuit Cookbook* for data converters**
- **Browse a complete list of amplifier and data converters circuits**

Visit ti.com/circuitcookbooks

Table of Contents

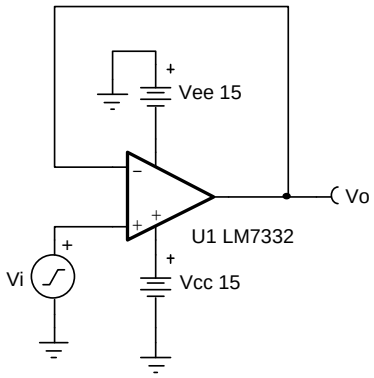
Buffer (follower) circuit

Design Goals

Input		Output		Freq.	Supply	
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	f	V_{cc}	V_{ee}
–10V	10V	–10V	10V	100kHz	15V	–15V

Design Description

This design is used to buffer signals by presenting a high input impedance and a low output impedance. This circuit is commonly used to drive low-impedance loads, analog-to-digital converters (ADC) and buffer reference voltages. The output voltage of this circuit is equal to the input voltage.



Design Notes

1. Use the op-amp linear output operating range, which is usually specified under the A_{OL} test conditions.
2. The small-signal bandwidth is determined by the unity-gain bandwidth of the amplifier.
3. Check the maximum output voltage swing versus frequency graph in the datasheet to minimize slew-induced distortion.
4. The common mode voltage is equal to the input signal.
5. Do not place capacitive loads directly on the output that are greater than the values recommended in the datasheet.
6. High output current amplifiers may be required if driving low impedance loads.
7. For more information on op-amp linear operating region, stability, slew-induced distortion, capacitive load drive, driving ADCs, and bandwidth, see the Design References section.

Design Steps

The transfer function for this circuit follows:

$$V_o = V_i$$

1. Verify that the amplifier can achieve the desired output swing using the supply voltages provided. Use the output swing stated in the A_{OL} test conditions. The output swing range of the amplifier must be greater than the output swing required for the design.

   $$-14V \le V_o \le 14V$$

   - The output swing of the LM7332 using ±15-V supplies is greater than the required output swing of the design. Therefore, this requirement is met.
   - Review the Output Voltage versus Output Current curves in the product datasheet to verify the desired output voltage can be achieved for the desired output current.

2. Verify the input common mode voltage of the amplifier will not be violated using the supply voltage provided. The input common mode voltage range of the amplifier must be greater than the input signal voltage range.

   $$-15.1\ V \le V_{icm} \le 15.1\ V$$

   - The input common-mode range of the LM7332 using ±15-V supplies is greater than the required input common-mode range of the design. Therefore, this requirement is met.

3. Calculate the minimum slew rate required to minimize slew-induced distortion.

   $$SR > 2 \times \pi \times Vp \times f = 2 \times \pi \times 10V \times 100kHz = 6.28V / \mu s$$

   - The slew rate of the LM7332 is 15.2V/µs. Therefore, this requirement is met.

4. Verify the device will have sufficient bandwidth for the desired output signal frequency.

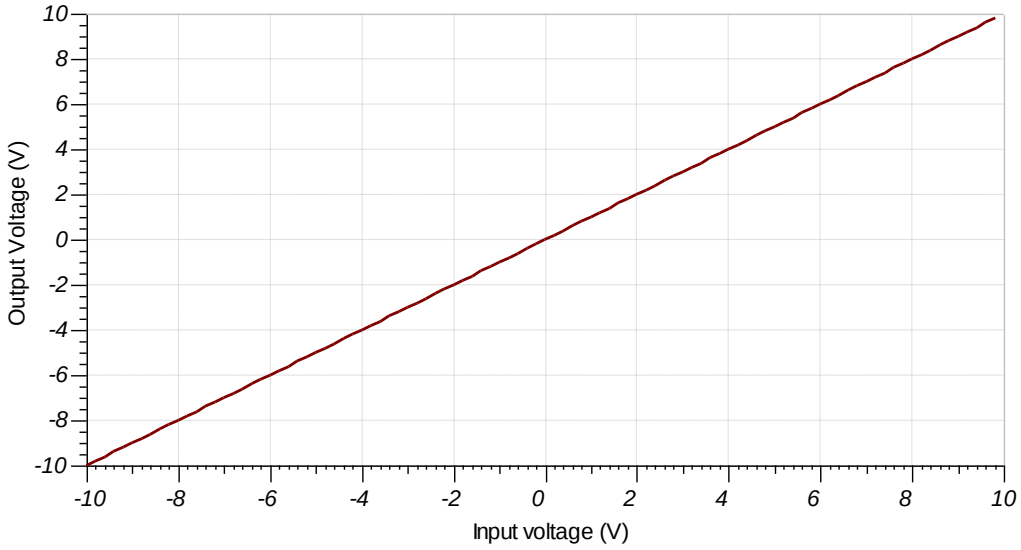
   $$f_{signal} < f_{unity}$$

   $$100kHz < 7.5MHz$$

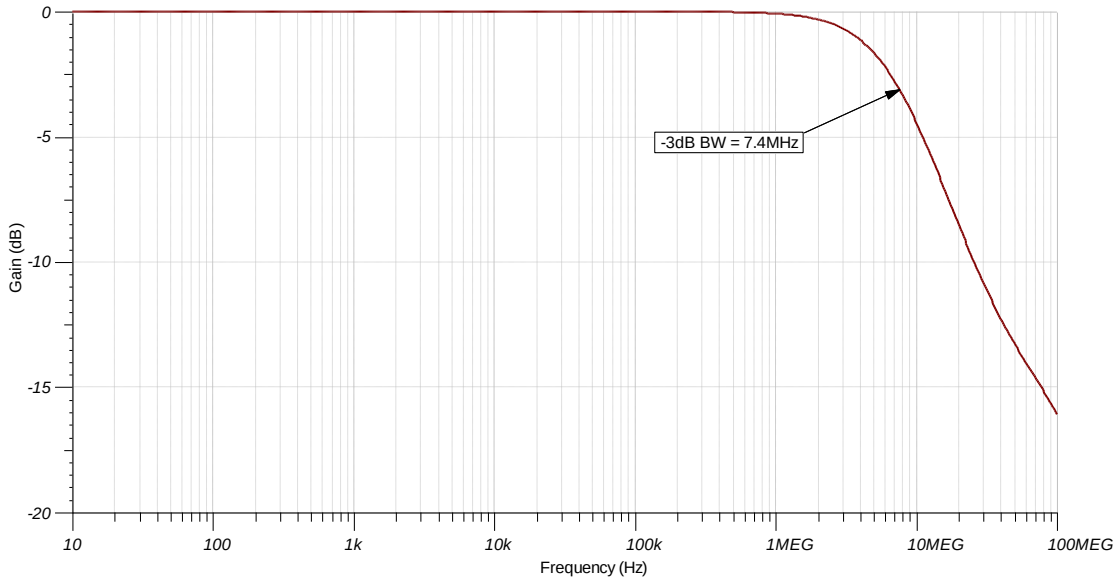
   - The desired output signal frequency is less than the unity-gain bandwidth of the LM7332. Therefore, this requirement is met.

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See the Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

For more information, see the Capacitive Load Drive Verified Reference Design Using an Isolation Resistor TI Design.

See the circuit SPICE simulation file SBOC491 – http://www.ti.com/lit/zip/sboc491.

For more information on many op amp topics including common-mode range, output swing, bandwidth, slew rate, and how to drive an ADC, see TI Precision Labs.

Design Featured Op Amp

LM7332	
V_{ss}	2.5V to 32V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1.6mV
I_q	2mA
I_b	1µA
UGBW	7.5MHz (±5-V supply)
SR	15.2V/µs
#Channels	2
www.ti.com/product/LM7332	

Design Alternate Op Amp

OPA192	
V_{ss}	4.5V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	1mA
I_b	5pA
UGBW	10MHz
SR	20V/µs
#Channels	1, 2, 4
www.ti.com/product/opa192	

The following device is for battery-operated or power-conscious designs outside of the original design goals described earlier, where lowering the total system power is desired.

LPV511	
V_{ss}	2.7V to 12V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.2mV
I_q	1.2µA
I_b	0.8nA
UGBW	27KHz
SR	7.5V/ms
#Channels	1
www.ti.com/product/lpv511	

Revision History

Revision	Date	Change
A	January 2019	Downscale title. Added LPV511 table in the Design Alternate Op Amp section.

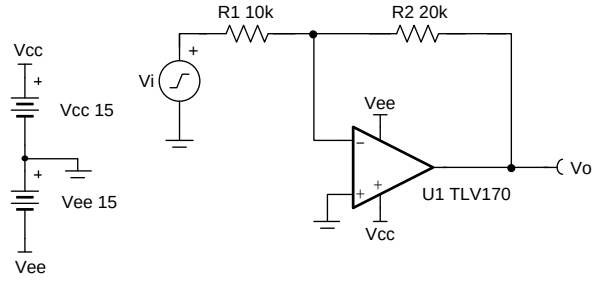
Inverting amplifier circuit

Design Goals

Input		Output		Freq.	Supply	
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	f	V_{cc}	V_{ee}
–7V	7V	–14V	14V	3kHz	15V	–15V

Design Description

This design inverts the input signal, V_i, and applies a signal gain of –2V/V. The input signal typically comes from a low-impedance source because the input impedance of this circuit is determined by the input resistor, R_1. The common-mode voltage of an inverting amplifier is equal to the voltage connected to the non-inverting node, which is ground in this design.



Design Notes

1. Use the op amp in a linear operating region. Linear output swing is usually specified under the A_{OL} test conditions. The common-mode voltage in this circuit does not vary with input voltage.
2. The input impedance is determined by the input resistor. Make sure this value is large when compared to the source's output impedance.
3. Using high value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
4. Avoid placing capacitive loads directly on the output of the amplifier to minimize stability issues.
5. Small-signal bandwidth is determined by the noise gain (or non-inverting gain) and op amp gain-bandwidth product (GBP). Additional filtering can be accomplished by adding a capacitor in parallel to R_2. Adding a capacitor in parallel with R_2 will also improve stability of the circuit if high value resistors are used.
6. Large signal performance may be limited by slew rate. Therefore, check the maximum output swing versus frequency plot in the data sheet to minimize slew-induced distortion.
7. For more information on op amp linear operating region, stability, slew-induced distortion, capacitive load drive, driving ADCs, and bandwidth please see the Design References section.

Design Steps

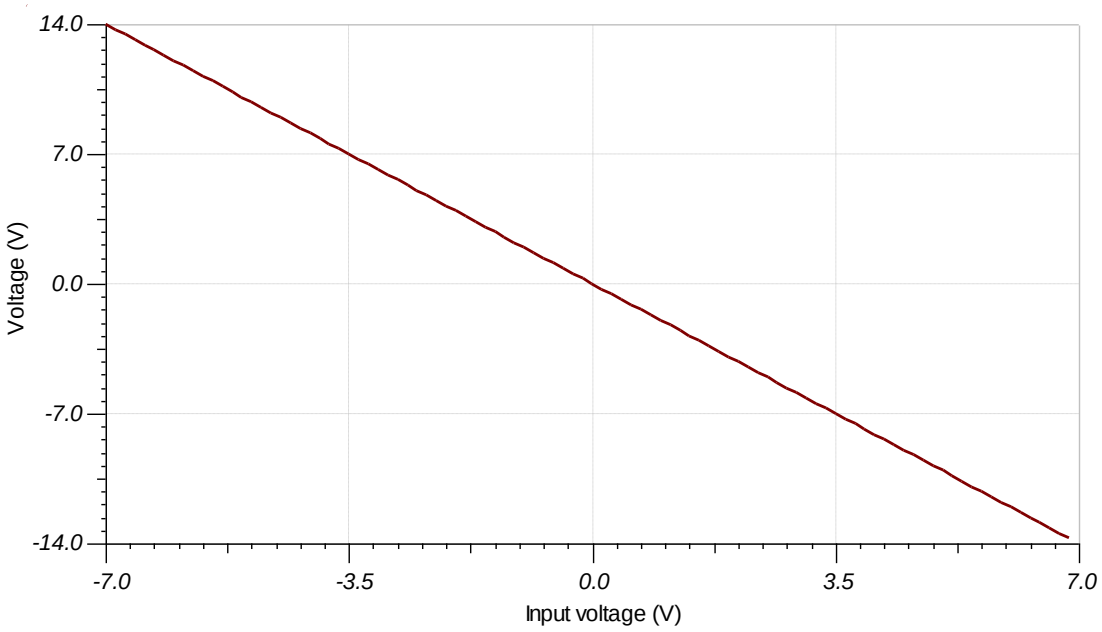
The transfer function of this circuit is given below.

$$V_o = V_i \times \left(-\frac{R_2}{R_1}\right)$$

1. Determine the starting value of R_1. The relative size of R_1 to the signal source's impedance affects the gain error. Assuming the signal source's impedance is low (for example, 100Ω), set R_1=10kΩ for 1% gain error.
   $$R_1 = 10k\Omega$$
2. Calculate the gain required for the circuit. Since this is an inverting amplifier use V_{iMin} and V_{oMax} for the calculation.
   $$G = \frac{V_{oMax}}{V_{iMin}} = \frac{14V}{-7V} = -2\frac{V}{V}$$
3. Calculate R_2 for a desired signal gain of –2V/V.
   $$G = -\frac{R_2}{R_1} \rightarrow R_2 = -G \times R_1 = -\left(-2\frac{V}{V}\right) \times 10k\Omega = 20k\Omega$$
4. Calculate the small signal circuit bandwidth to ensure it meets the 3kHz requirement. Be sure to use the noise gain, or non-inverting gain, of the circuit.
   $$GBP_{TLV170} = 1.2MHz \quad ($$
   $$NG = 1 + \frac{R_2}{R_1} = 3\frac{V}{V}$$
   $$BW = \frac{GBP}{NG} = \frac{1.2MHz}{3V/V} = 400kHz$$
5. Calculate the minimum slew rate required to minimize slew-induced distortion.
   $$V_p = \frac{SR}{2 \times \pi \times f} \rightarrow SR > 2 \times \pi \times f \times V_p$$
   $$SR > 2 \times \pi \times 3kHz \times 14V = 263.89\frac{kV}{s} = 0.26\frac{V}{\mu s}$$
   - SR_{TLV170}=0.4V/µs, therefore it meets this requirement.
6. To avoid stability issues ensure that the zero created by the gain setting resistors and input capacitance of the device is greater than the bandwidth of the circuit.
   $$\frac{1}{2 \times \pi \times (C_{cm} + C_{diff}) \times (R_2 \parallel R_1)} > \frac{GBP}{NG}$$
   $$\frac{1}{2 \times \pi \times \ 3pF + 3pF \ \times \frac{20k\Omega \times 10k\Omega}{20k\Omega + 10k\Omega}} > \frac{1.2MHz}{3V/V}$$
   $$43.77MHz > 400kHz$$
   - C_{cm} and C_{diff} are the common-mode and differential input capacitances of the TLV170, respectively.
   - Since the zero frequency is greater than the bandwidth of the circuit, this requirement is met.

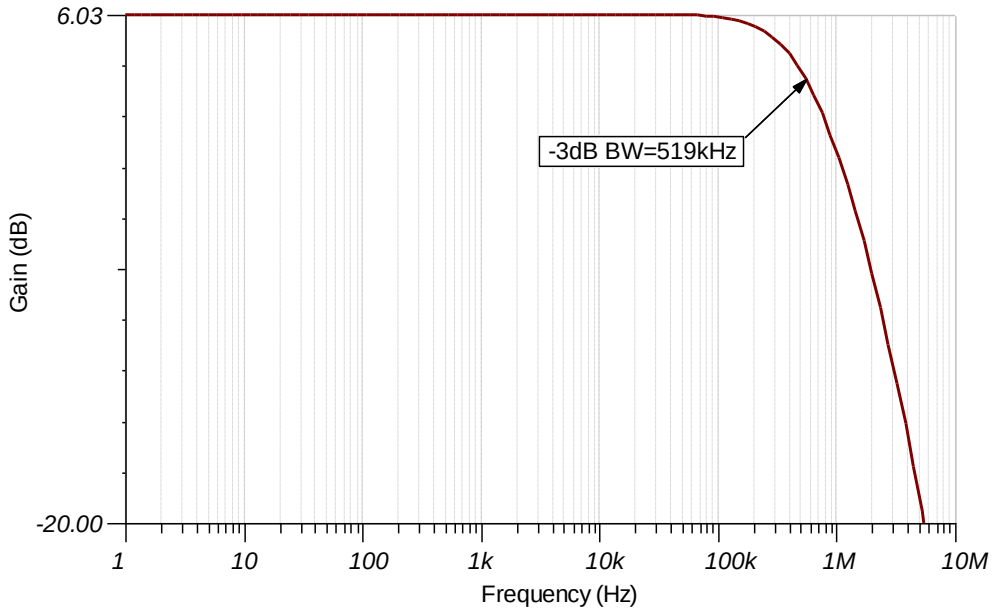
Design Simulations

DC Simulation Results



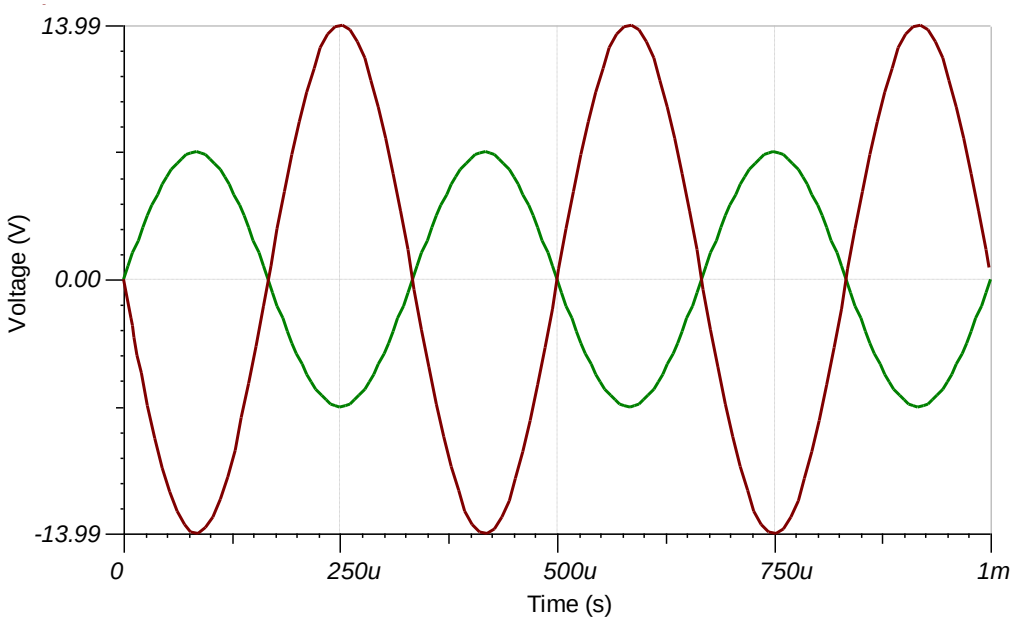
AC Simulation Results

The bandwidth of the circuit depends on the noise gain, which is 3V/V. The bandwidth is determined by looking at the –3dB point, which is located at 3dB given a signal gain of 6dB. The simulation sufficiently correlates with the calculated value of 400kHz.



Transient Simulation Results

The output is double the magnitude of the input, and inverted.



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC492.

For more information on many op amp topics including common-mode range, output swing, bandwidth, and how to drive an ADC please visit TI Precision Labs.

Design Featured Op Amp

TLV170	
V_{ss}	±18V (36V)
V_{inCM}	(Vee-0.1V) to (Vcc-2V)
V_{out}	Rail-to-rail
V_{os}	0.5mV
I_q	125µA
I_b	10pA
UGBW	1.2MHz
SR	0.4V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv170	

Design Alternate Op Amp

LMV358	
V_{ss}	2.7 to 5.5V
V_{inCM}	(V_{ee}–0.2V) to (V_{cc}–0.8V)
V_{out}	Rail-to-rail
V_{os}	1.7mV
I_q	210µA
I_b	15nA
UGBW	1MHz
SR	1V/µs
#Channels	1 (LMV321), 2 (LMV358), 4 (LMV324)
www.ti.com/product/lmv358	

Revision History

Revision	Date	Change
A	January 2019	Downscale title. Added link to circuit cookbook landing page.

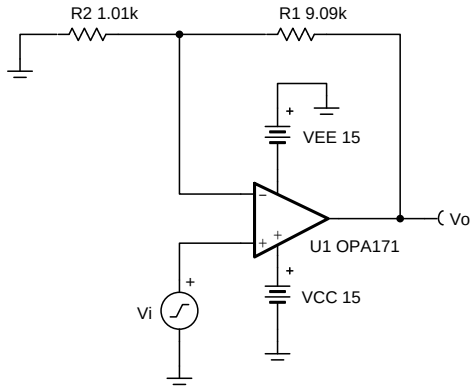
Non-inverting amplifier circuit

Design Goals

Input		Output		Supply	
ViMin	ViMax	VoMin	VoMax	Vcc	Vee
–1V	1V	–10V	10	15V	–15V

Design Description

This design amplifies the input signal, V_i, with a signal gain of 10V/V. The input signal may come from a high-impedance source (for example, MΩ) because the input impedance of this circuit is determined by the extremely high input impedance of the op amp (for example, GΩ). The common-mode voltage of a non-inverting amplifier is equal to the input signal.



Design Notes

1. Use the op amp linear output operating range, which is usually specified under the A_{OL} test conditions. The common-mode voltage is equal to the input signal.
2. The input impedance of this circuit is equal to the input impedance of the amplifier.
3. Using high-value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
4. Avoid placing capacitive loads directly on the output of the amplifier to minimize stability issues.
5. The small-signal bandwidth of a non-inverting amplifier depends on the gain of the circuit and the gain bandwidth product (GBP) of the amplifier. Additional filtering can be accomplished by adding a capacitor in parallel to R_1. Adding a capacitor in parallel with R_1 will also improve stability of the circuit if high-value resistors are used.
6. Large signal performance may be limited by slew rate. Therefore, check the maximum output swing versus frequency plot in the data sheet to minimize slew-induced distortion.
7. For more information on op amp linear operating region, stability, slew-induced distortion, capacitive load drive, driving ADCs, and bandwidth please see the Design References section.

Design Steps

The transfer function for this circuit is given below.

$$V_o = V_i \times \left(1 + \frac{R_1}{R_2}\right)$$

1. Calculate the gain.

$$G = \frac{V_{o\_max} - V_{o\_min}}{V_{i\_max} - V_{i\_min}} \quad (\text{(}$$

$$G = \frac{10V - \ -10V}{1\ V - \ -1\ V} = 10V / V$$

2. Calculate values for R_1 and R_2.

$$G = 1 + \frac{R_1}{R_2}$$

Choose $R_1 = 9.09k\Omega$

$$R_2 = \frac{R_1}{G-1} = \frac{9.09k\Omega}{10V/V \ -1} = 1.01k\Omega$$

3. Calculate the minimum slew rate required to minimize slew-induced distortion.

$$SR > 2 \times \pi \times V_p \times f = 2 \times \pi \times 10V \times 20kHz = 1.257V / \mu s$$

- The slew rate of the OPA171 is 1.5V/µs, therefore it meets this requirement.

4. To maintain sufficient phase margin, ensure that the zero created by the gain setting resistors and input capacitance of the device is greater than the bandwidth of the circuit.

$$\frac{1}{2 \times \pi \times (C_{cm} + C_{diff}) \times (R_1 \parallel R_2)} > \frac{GBP}{G} \quad (\text{(}$$

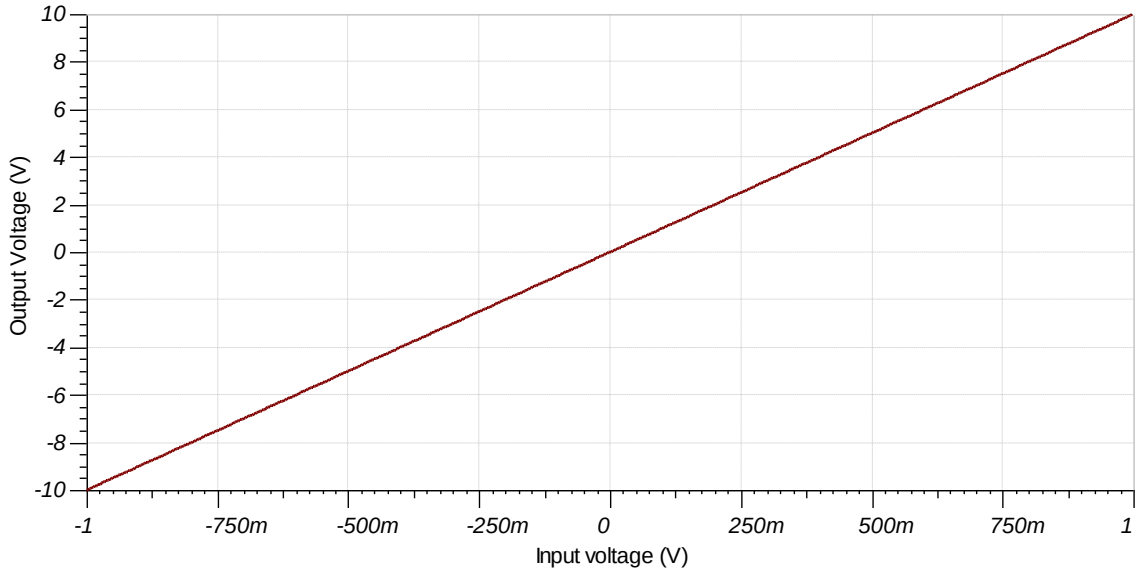
$$\frac{1}{2 \times \pi \times \ 3pF + 3pF \ \times \frac{1.01k\Omega \times 9.09k\Omega}{1.01k\Omega + 9.09k\Omega}} > \frac{3MHz}{10V/V}$$

$$29.18MHz > 300kHz$$

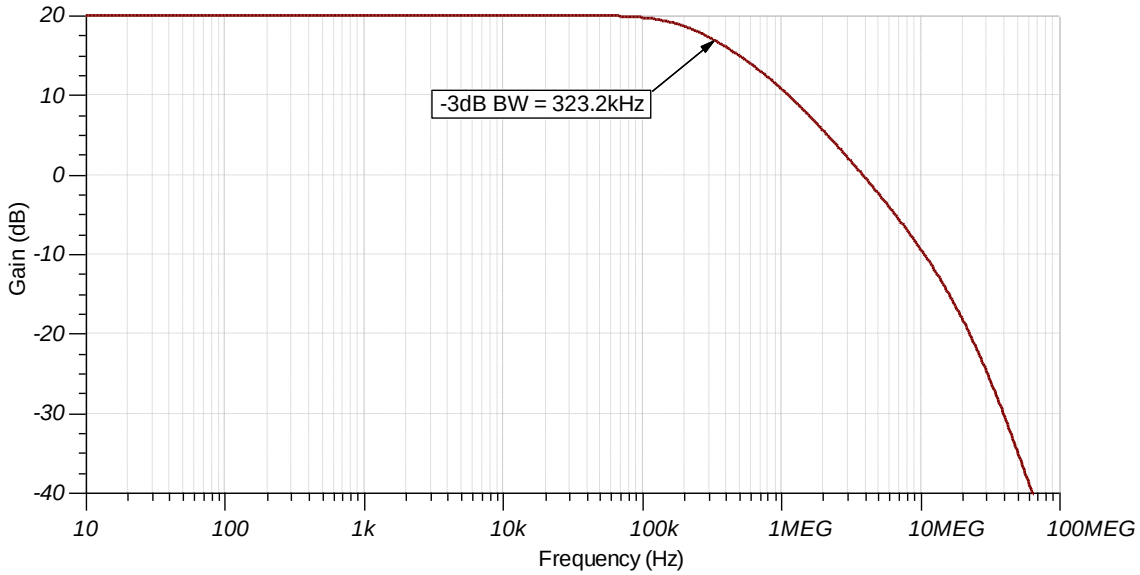
- C_{cm} and C_{diff} are the common-mode and differential input capacitances of the OPA171, respectively.
- Since the zero frequency is greater than the bandwidth of the circuit, this requirement is met.

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC493.

For more information on many op amp topics including common-mode range, output swing, and bandwidth please visit TI Precision Labs.

Design Featured Op Amp

OPA171	
V_{ss}	2.7V to 36V
V_{inCM}	(V_{ee}–0.1V) to (V_{cc}–2V)
V_{out}	Rail-to-rail
V_{os}	250µV
I_q	475µA
I_b	8pA
UGBW	3MHz
SR	1.5V/µs
#Channels	1, 2, 4
www.ti.com/product/opa171	

Design Alternate Op Amp

OPA191	
V_{ss}	4.5V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	140µA
I_b	5pA
UGBW	2.5MHz
SR	7.5V/µs
#Channels	1, 2, 4
www.ti.com/product/OPA191	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

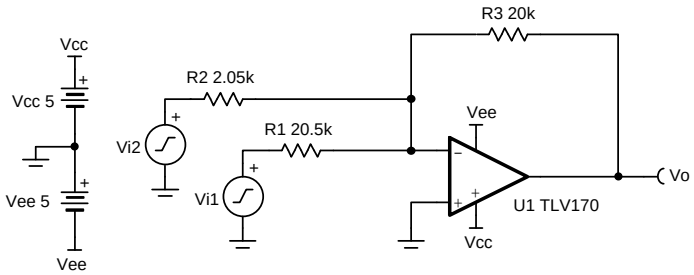
Inverting summer circuit

Design Goals

Input 1		Input 2		Output		Freq.	Supply	
V_{i1Min}	V_{i1Max}	V_{i2Min}	V_{i2Max}	V_{oMin}	V_{oMax}	f	V_{cc}	V_{ee}
–5V	5V	–250mV	250mV	–4.9V	4.9V	10kHz	5V	–5V

Design Description

This design sums (adds) and inverts two input signals, V_{i1} and V_{i2}. The input signals typically come from low-impedance sources because the input impedance of this circuit is determined by the input resistors, R_1 and R_2. The common-mode voltage of an inverting amplifier is equal to the voltage connected to the non-inverting node, which is ground in this design.



Design Notes

1. Use the op amp in a linear operating region. Linear output swing is usually specified under the A_{OL} test conditions. The common-mode voltage in this circuit does not vary with input voltage.
2. The input impedance is determined by the input resistors. Make sure these values are large when compared to the output impedance of the source.
3. Using high-value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
4. Avoid placing capacitive loads directly on the output of the amplifier to minimize stability issues.
5. Small-signal bandwidth is determined by the noise gain (or non-inverting gain) and op amp gain-bandwidth product (GBP). Additional filtering can be accomplished by adding a capacitor in parallel to R_3. Adding a capacitor in parallel with R_3 will also improve stability of the circuit if high-value resistors are used.
6. Large signal performance may be limited by slew rate. Therefore, check the maximum output swing versus frequency plot in the data sheet to minimize slew-induced distortion.
7. For more information on op amp linear operating region, stability, slew-induced distortion, capacitive load drive, driving ADCs, and bandwidth please see the Design References section.

Design Steps

The transfer function for this circuit is given below.

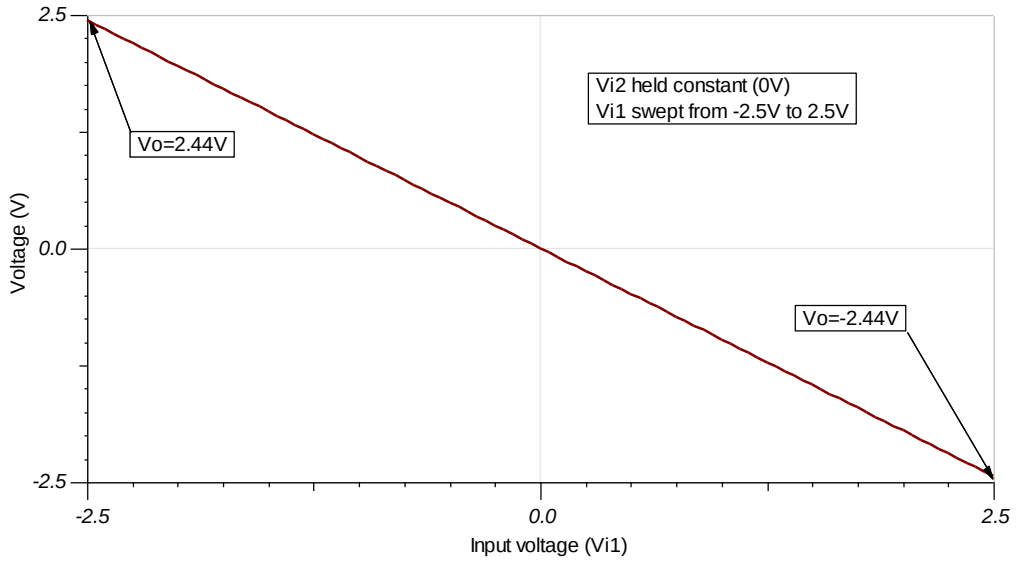
$$V_o = V_{i\,1} \times \left(-\frac{R_3}{R_1}\right) + V_{i\,2} \times \left(-\frac{R_3}{R_2}\right)$$

1. Select a reasonable resistance value for R_3.
   $R_3 = 20k\Omega$
2. Calculate gain required for V_{i1}. For this design, half of the output swing is devoted to each input.
   $$G_{Vi1} = \left|\frac{\frac{V_{oMax} - V_{oMin}}{2}}{V_{i\,1\,Max} - V_{i\,1\,Min}}\right| = \left|\frac{\frac{4.9V - (-4.9V)}{2}}{2.5V - (-2.5V)}\right| = 0.98\frac{V}{V} = -0.175dB$$
3. Calculate the value of R_1.
   $$|G_{Vi1}| = \frac{R_3}{R_1} \rightarrow R_1 = \frac{R_3}{|G_{Vi1}|} = \frac{20k\Omega}{0.98V/V} = 20.4k\Omega \approx 20.5k\Omega \text{ (Standard Value)}$$
4. Calculate gain required for V_{i2}. For this design, half of the output swing is devoted to each input.
   $$G_{Vi2} = \left|\frac{\frac{V_{oMax} - V_{oMin}}{2}}{V_{i\,2\,Max} - V_{i\,2\,Min}}\right| = \left|\frac{\frac{4.9V - (-4.9V)}{2}}{250mV - (-250mV)}\right| = 9.8\frac{V}{V} = 19.82dB$$
5. Calculate the value of R_2.
   $$|G_{Vi2}| = \frac{R_3}{R_2} \rightarrow R_2 = \frac{R_3}{|G_{Vi2}|} = \frac{20k\Omega}{9.8V/V} = 2.04k\Omega \approx 2.05k\Omega \text{ (Standard Value)}$$
6. Calculate the small signal circuit bandwidth to ensure it meets the 10-kHz requirement. Be sure to use the noise gain (NG), or non-inverting gain, of the circuit. When calculating the noise gain note that R_1 and R_2 are in parallel.
   $$GBP_{OPA170} = 1.2MHz \quad (\quad) \quad (\quad)$$
   $$NG = 1 + \frac{R_3}{R_1 \| R_2} = 1 + \frac{20k\Omega}{1.86k\Omega} = 11.75\frac{V}{V} = 21.4dB$$
   $$BW = \frac{GBP}{NG} = \frac{1.2MHz}{11.75V/V} = 102kHz$$
   - This requirement is met because the closed-loop bandwidth is 102kHz and the design goal is 10kHz.
7. Calculate the minmum slew rate to minimize slew-induced distortion.
   $$V_p = \frac{SR}{2 \times \pi \times f} \rightarrow SR > 2 \times \pi \times f \times V_p$$
   $$SR > 2 \times \pi \times 10kHz \times 4.9V = 307.87\frac{kV}{s} = 0.31\frac{V}{\mu s}$$
   - SR_{OPA170}=0.4V/µs, therefore it meets this requirement.
8. To avoid stability issues ensure that the zero created by the gain setting resistors and input capacitance of the device is greater than the bandwidth of the circuit.
   $$\frac{\frac{1}{2 \times \pi \times (C_{cm} + C_{diff}) \times (R_1 \| R_2 \| R_3)}}{\frac{1}{2 \times \pi \times \quad 3pF + 3pF \quad \times 1.7k\Omega}} > \frac{\frac{GBP}{NG}}{\frac{1.2MHz}{11.75V/V}}$$
   $$15.6MHz > 102kHz$$
   - C_{cm} and C_{diff} are the common-mode and differential input capacitances.
   - Since the zero frequency is greater than the bandwidth of the circuit, this requirement is met.

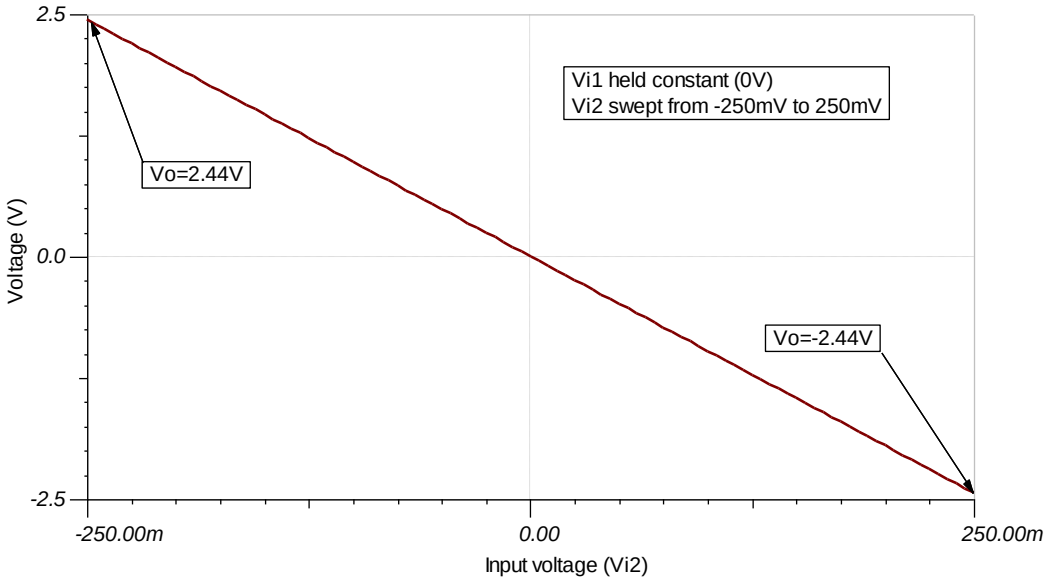
Design Simulations

DC Simulation Results

This simulation sweeps V_{i1} from –2.5V to 2.5V while V_{i2} is held constant at 0V. The output is inverted and ranges from –2.44V to 2.44V.

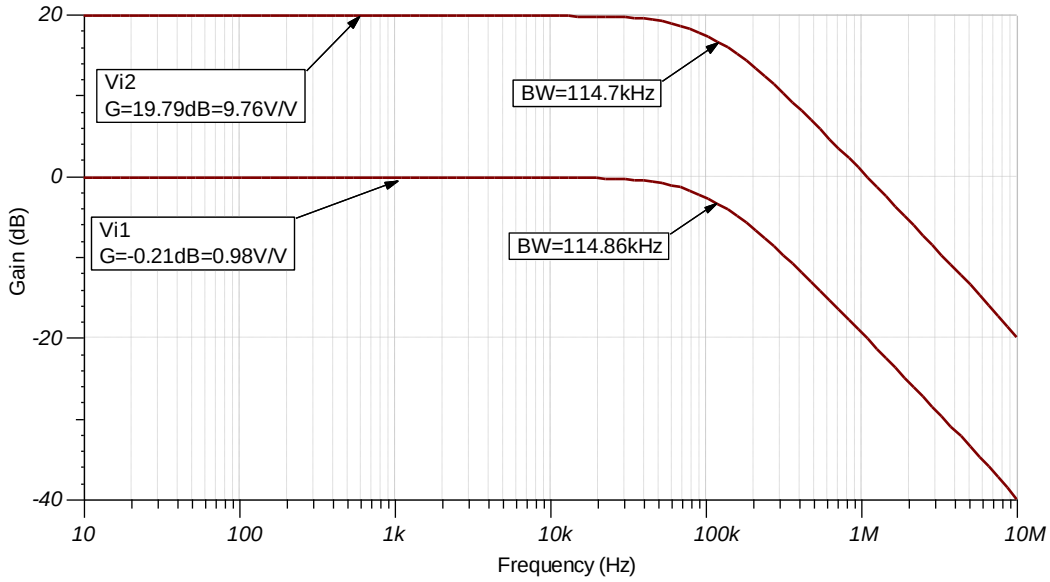


This simulation sweeps V_{i2} from –250mV to 250mV while V_{i1} is held constant at 0V. The output is inverted and ranges from –2.44V to 2.44V.



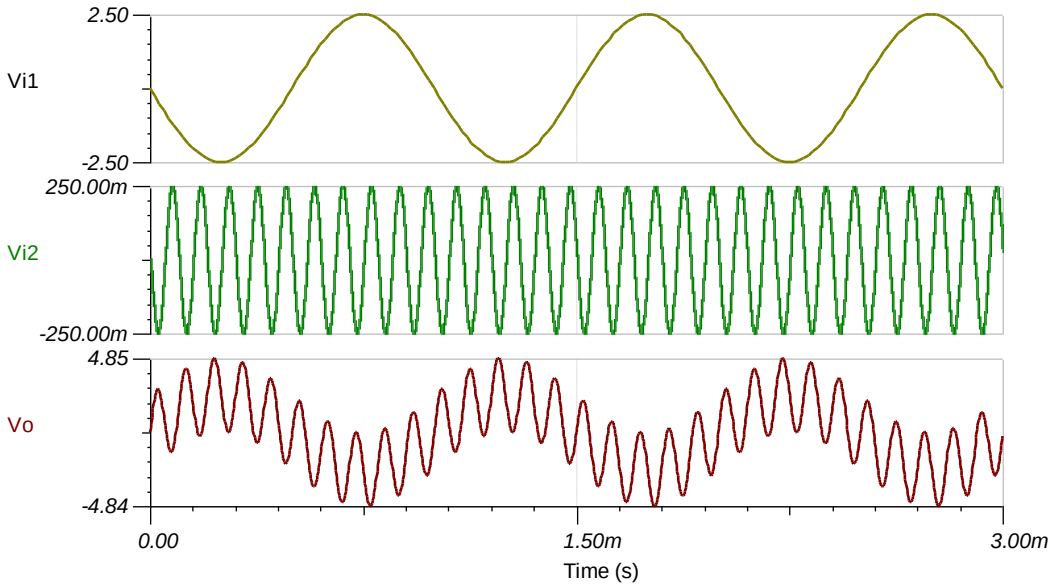
AC Simulation Results

This simulation shows the bandwidth of the circuit. Note that the bandwidth is the same for either input. This is because the bandwidth depends on the noise gain of the circuit, not the signal gain of each input. These results correlate well with the calculations.



Transient Simulation Results

This simulation shows the inversion and summing of the two input signals. V_{i1} is a 1-kHz, 5-V_{pp} sine wave and V_{i2} is a 10-kHz, 500-mV_{pp} sine wave. Since both inputs are properly amplified or attenuated, the output is within specification.



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC494.

For more information on many op amp topics including common-mode range, output swing, bandwidth, and how to drive an ADC please visit TI Precision Labs.

Design Featured Op Amp

OPA170	
V_{ss}	2.7V to 36V
V_{inCM}	(Vee-0.1V) to (Vcc-2V)
V_{out}	Rail-to-rail
V_{os}	0.25mV
I_q	110µA
I_b	8pA
UGBW	1.2MHz
SR	0.4V/µs
#Channels	1, 2, 4
www.ti.com/product/opa170	

Design Alternate Op Amp

LMC7101	
V_{ss}	2.7V to 15.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	110µV
I_q	0.8mA
I_b	1pA
UGBW	1.1MHz
SR	1.1V/µs
#Channels	1
www.ti.com/product/lmc7101	

Revision History

Revision	Date	Change
A	January 2019	Downscale title. Updated title role to 'Amplifiers'. Added link to circuit cookbook landing page.

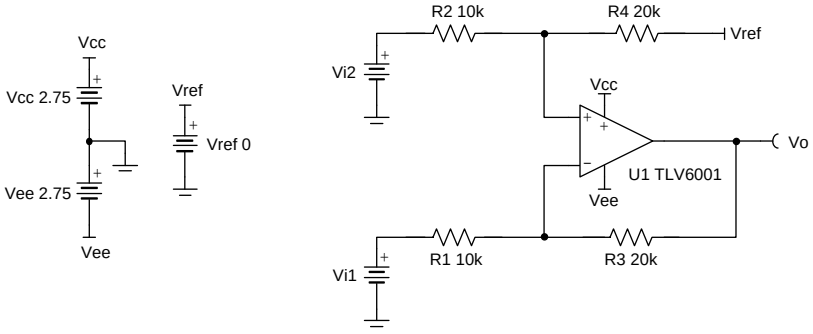
Difference amplifier (subtractor) circuit

Design Goals

Input (V_{i2}-V_{i1})		Output		CMRR (min)	Supply		
$V_{idiffMin}$	$V_{idiffMax}$	V_{oMin}	V_{oMax}	dB	V_{cc}	V_{ee}	V_{ref}
–1.25V	1.25V	–2.5V	2.5V	50	2.75V	–2.75V	0V

Design Description

This design inputs two signals, V_{i1} and V_{i2}, and outputs their difference (subtracts). The input signals typically come from low-impedance sources because the input impedance of this circuit is determined by the resistive network. Difference amplifiers are typically used to amplify differential input signals and reject common-mode voltages. A common-mode voltage is the voltage common to both inputs. The effectiveness of the ability of a difference amplifier to reject a common-mode signal is known as common-mode rejection ratio (CMRR). The CMRR of a difference amplifier is dominated by the tolerance of the resistors.



Design Notes

1. Use the op amp in a linear operating region. Ensure that the inputs of the op amp do not exceed the common-mode range of the device. Linear output swing is usually specified under the A_{OL} test conditions.
2. The input impedance is determined by the input resistive network. Make sure these values are large when compared to the output impedance of the sources.
3. Using high-value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
4. Avoid placing capacitive loads directly on the output of the amplifier to minimize stability issues.
5. Small-signal bandwidth is determined by the noise gain (or non-inverting gain) and op amp gain-bandwidth product (GBP). Additional filtering can be accomplished by adding a capacitors in parallel to R_3 and R_4. Adding capacitors in parallel with R_3 and R_4 will also improve stability of the circuit if high-value resistors are used.
6. Large signal performance may be limited by slew rate. Therefore, check the maximum output swing versus frequency plot in the data sheet to minimize slew-induced distortion.
7. For more information on op amp linear operating region, stability, slew-induced distortion, capacitive load drive, driving ADCs, and bandwidth please see the Design References section.

Design Steps

The complete transfer function for this circuit is shown below.

$$V_o = V_{i1} \times \left(-\frac{R_3}{R_1}\right) + V_{i2} \times \left(\frac{R_4}{R_2+R_4}\right) \times \left(1+\frac{R_3}{R_1}\right) + Vref \times \left(\frac{R_2}{R_2+R_4}\right) \times \left(1+\frac{R_3}{R_1}\right)$$

If $R_1 = R_2$ and $R_3 = R_4$ the transfer function for this circuit simplifies to the following equation.

$$V_o = (V_{i2} - V_{i1}) \times \frac{R_3}{R_1} + Vref$$

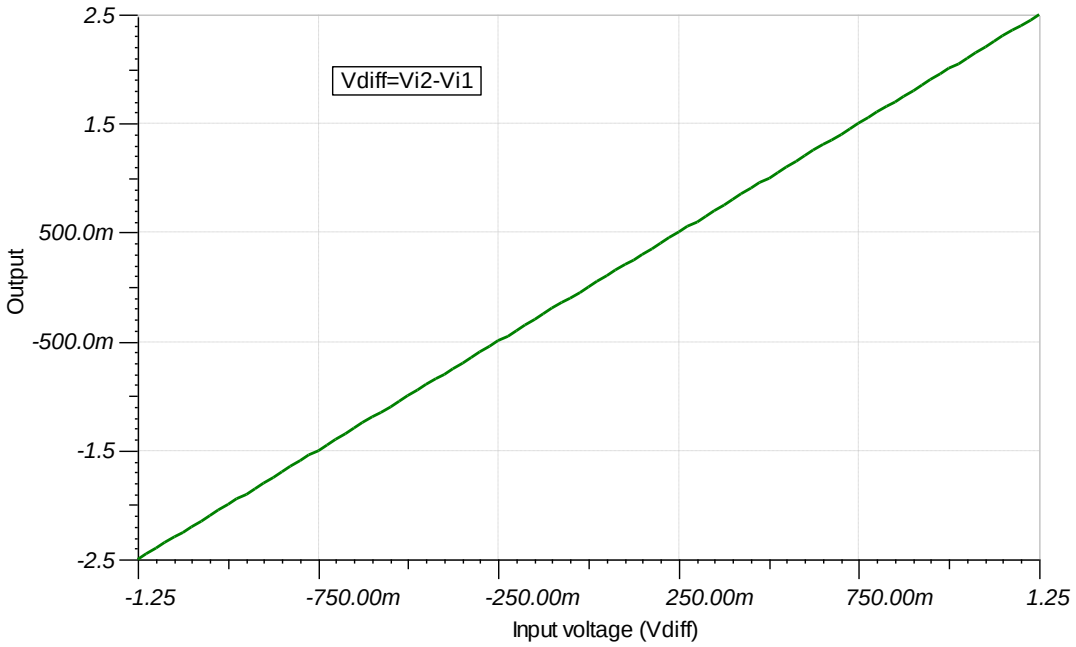
- Where the gain, G, is R_3/R_1.

1. Determine the starting value of R_1 and R_2. The relative size of R_1 and R_2 to the signal impedance of the source affects the gain error.
   $$R_1 = R_2 = 10k\Omega$$
2. Calculate the gain required for the circuit.
   $$G = \frac{V_{oMax} - V_{oMin}}{V_{idiffMax} - V_{idiffMin}} = \frac{2.5V-(-2.5V)}{1.25V-(-1.25V)} = 2\frac{V}{V} = 6.02dB$$
3. Calculate the values for R_3 and R_4.
   $$G = 2\frac{V}{V} = \frac{R_3}{R_1} \rightarrow 2 \times R_1 = R_3 = R_4 = 20k\Omega$$
4. Calculate resistor tolerance to meet the minimum common-mode rejection ratio (CMRR). For minimum (worst-case) CMRR, α = 4. For a more probable, or typical value of CMRR, α = 0.33.
   $$CMRR_{dB} \cong 20\log10\left(\frac{1+G}{\alpha \times \varepsilon}\right)$$
   ( ) ( )
   $$\varepsilon = \frac{1+G}{\alpha \times 10^{\frac{CMRR_{dB}}{20}}} = \frac{3}{4 \times 10^{\frac{50}{20}}} = 0.024 = 0.24\% \rightarrow \text{Use } 0.1 \text{ \% resistors}$$
5. For quick reference, the following table compares resistor tolerance to minimum and typical CMRR values assuming G = 1 or G = 2. As shown above, as gain increases so does CMRR.

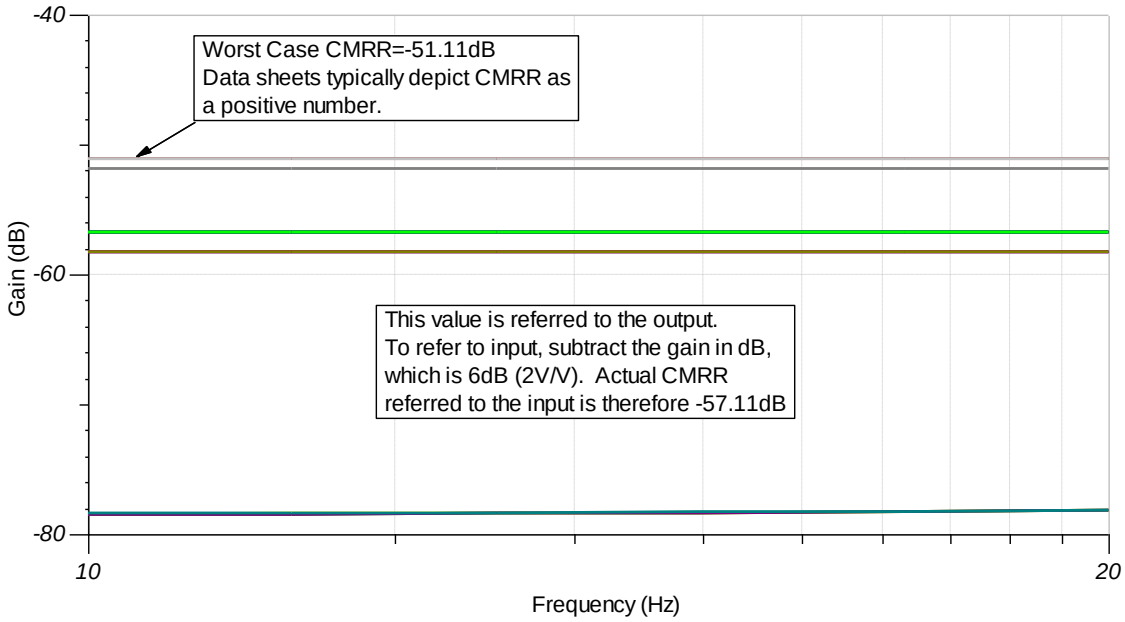
Tolerance	G=1 Minimum (dB)	G=1 Typical (dB)	G=2 Minimum (dB)	G=2 Typical (dB)
0.01%=0.0001	74	95.6	77.5	99.2
0.1%=0.001	54	75.6	57.5	79.2
0.5%=0.005	40	61.6	43.5	65.2
1%=0.01	34	55.6	37.5	59.2
5%=0.05	20	41.6	23.5	45.2

Design Simulations

DC Simulation Results



CMRR Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC495.

For more information on many op amp topics including common-mode range, output swing, bandwidth, and how to drive an ADC please visit TI Precision Labs. For more information on difference amplifier CMRR, please read Overlooking the obvious: the input impedance of a difference amplifier .

Design Featured Op Amp

TLV6001	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	750µV
I_q	75µA
I_b	1pA
UGBW	1MHz
SR	0.5V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv6001	

Design Alternate Op Amp

OPA320	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	40µV
I_q	1.5mA
I_b	0.2pA
UGBW	20MHz
SR	10V/µs
#Channels	1, 2
www.ti.com/product/opa320	

Revision History

Revision	Date	Change
A	January 2019	Downscale title. Added link to circuit cookbook landing page.

Two op amp instrumentation amplifier circuit

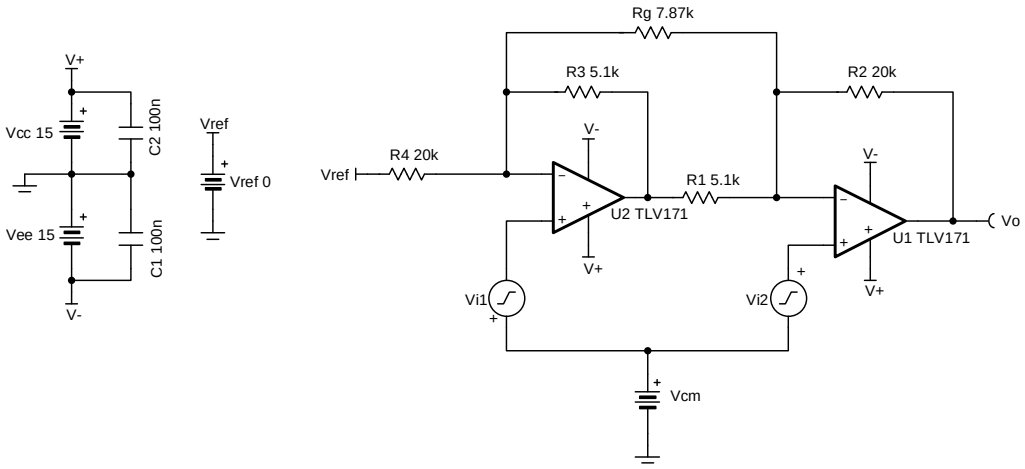
Design Goals

Input $V_{iDiff}(V_{i2} - V_{i1})$		Output		Supply		
V_{iDiff_Min}	V_{iDiff_Max}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
+/–1V	+/–2V	–10V	+10V	15V	–15V	0V

V_{cm}	Gain Range
+/-10V	5V/V to 10V/V

Design Description

This design amplifiers the difference between V_{i1} and V_{i2} and outputs a single ended signal while rejecting the common–mode voltage. Linear operation of an instrumentation amplifier depends upon the linear operation of its primary building block: op amps. An op amp operates linearly when the input and output signals are within the device's input common–mode and output–swing ranges, respectively. The supply voltages used to power the op amps define these ranges.



Design Notes

1. R_g sets the gain of the circuit.
2. High–value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
3. The ratio of R_4 and R_3 set the minimum gain when R_g is removed.
4. Ratios of R_2/R_1 and R_4/R_3 must be matched to avoid degrading the instrumentation amplifier's DC CMRR and ensuring the V_{ref} gain is 1V/V.
5. Linear operation is contingent upon the input common–mode and the output swing ranges of the discrete op amps used. The linear output swing ranges are specified under the A_{ol} test conditions in the op amps datasheets.

Design Steps

1. Transfer function of this circuit.

   $V_o = V_{iDiff} \times G + V_{ref} = (V_{i2} - V_{i1}) \times G + V_{ref}$

   when $V_{ref} = 0$, the transfer function simplifies to the following equation:

   $V_o = (V_{i2} - V_{i1}) \times G$

   where G is the gain of the instrumentation amplifier and $G = 1 + \frac{R_4}{R_3} + \frac{2R_2}{R_g}$

2. Select R_4 and R_3 to set the minimum gain.

   $G_{min} = 1 + \frac{R_4}{R_3} = 5\frac{V}{V}$

   Choose $R_4 = 20k\Omega$

   $G_{min} = 1 + \frac{20k\Omega}{R_3} = 5\frac{V}{V}$

   $R_3 = \frac{R_4}{5-1} = \frac{20k\Omega}{4} = 5k\Omega \rightarrow R_3 = 5.1k\Omega$ (Standard Value)

3. Select R_1 and R_2. Ensure that R_1/R_2 and R_3/R_4 ratios are matched to set the gain applied to the reference voltage at 1V/V.

   $\frac{V_{o\_ref}}{Vref} = (-\frac{R_3}{R_4}) \times (-\frac{R_2}{R_1}) = \frac{R_3 \times R_2}{R_4 \times R_1} = 1\frac{V}{V}$

   $\frac{R_2}{R_1} = \frac{R_4}{R_3} \rightarrow R_1 = R_3 = 5.1k\Omega$ and $R_2 = R_4 = 20k\Omega$ (Standad Value)

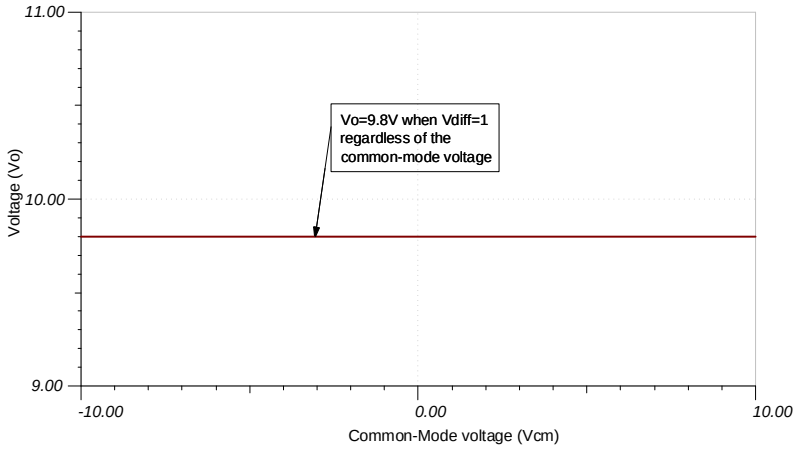
4. Select R_g to meet the desired maximum gain G = 10V/V.

   $G = 1 + \frac{R_4}{R_3} + \frac{2R_2}{R_g} = 1 + \frac{20\ k\Omega}{5.1\ k\Omega} + \frac{2 \times 20\ k\Omega}{R_g} = 10\ V/V$

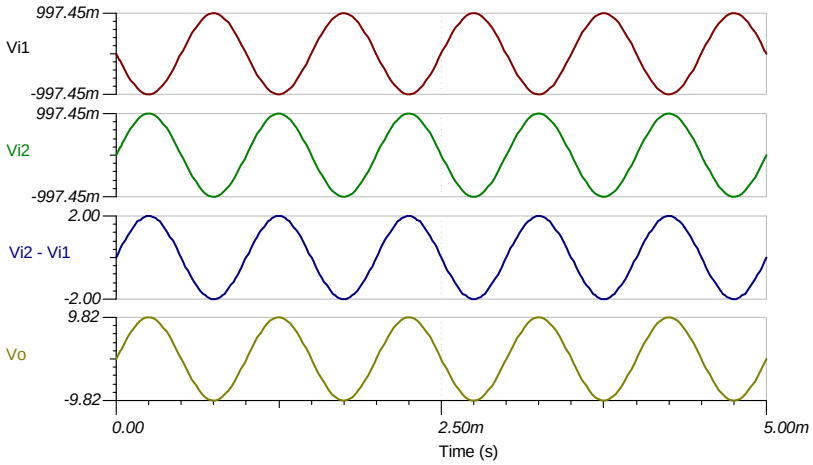
   $R_g = 8\ k\Omega \rightarrow R_g = 7.87\ k\Omega$ (Standard Value)

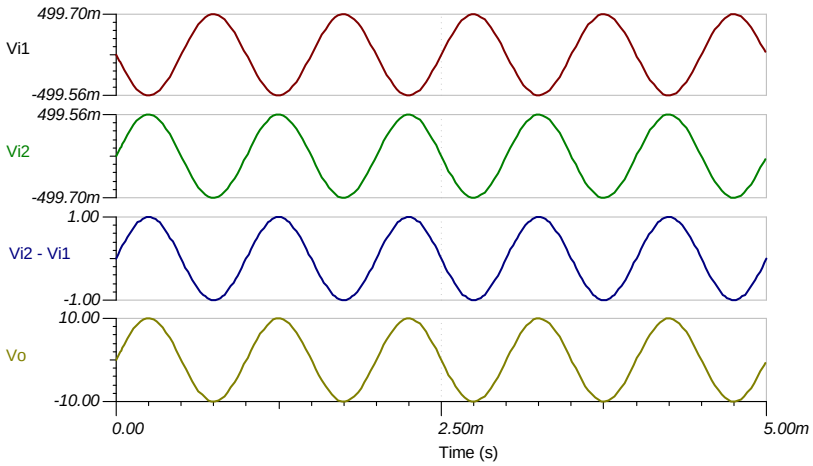
Design Simulations

DC Simulation Results



Transient Simulation Results





References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAU7
3. TI Precision Labs
4. V_{CM} vs. V_{OUT} plots for instrumentation amplifiers with two op amps
5. Common-mode Range Calculator for Instrumentation Amplifiers

Design Featured Op Amp

TLV171	
V_{ss}	4.5V to 36V
V_{inCM}	(V_{ee}–0.1V) to (V_{cc}–2V)
V_{out}	Rail–to–rail
V_{os}	0.25mV
I_q	475µA
I_b	8pA
UGBW	3MHz
SR	1.5V/µs
#Channels	1,2,4
www.ti.com/product/tlv171	

Design Alternate Op Amp

OPA172	
V_{ss}	4.5V to 36V
V_{inCM}	(V_{ee}–0.1V) to (V_{cc}–2V)
V_{out}	Rail–to–rail
V_{os}	0.2mV
I_q	1.6mA
I_b	8pA
UGBW	10MHz
SR	10V/µs
#Channels	1,2,4
www.ti.com/product/opa172	

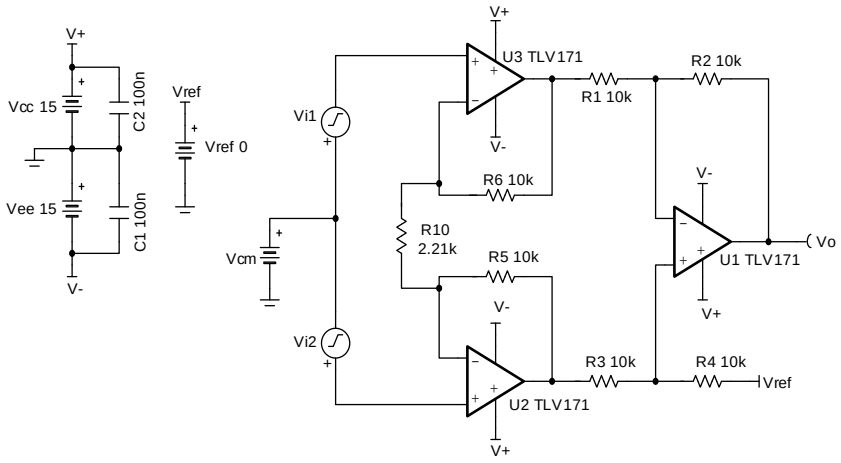
Three op amp instrumentation amplifier circuit

Design Goals

Input V_{idiff} (V_{i2} - V_{i1})		Common-mode Voltage	Output		Supply		
$V_{i\ diff\ Min}$	$V_{i\ diff\ Max}$	V_{cm}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
-0.5V	+0.5V	±7V	–5V	+5V	+15V	–15V	0V

Design Description

This design uses 3 op amps to build a discrete instrumentation amplifier. The circuit converts a differential signal to a single-ended output signal. Linear operation of an instrumentation amplifier depends upon linear operation of its building block: op amps. An op amp operates linearly when the input and output signals are within the device's input common-mode and output swing ranges, respectively. The supply voltages used to power the op amps define these ranges.



Design Notes

1. Use precision resistors to achieve high DC CMRR performance
2. R_{10} sets the gain of the circuit.
3. Add an isolation resistor to the output stage to drive large capacitive loads.
4. High-value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
5. Linear operation is contingent upon the input common-mode and the output swing ranges of the discrete op amps used. The linear output swing ranges are specified under the A_{ol} test conditions in the op amps datasheets.

Design Steps

1. Transfer function of this circuit:

$$V_o = (V_{i2} - V_{i1}) \times G + V_{ref}$$

When $V_{ref} = 0$, the transfer function simplifies to the following equation:

$$V_o = (V_{i2} - V_{i1}) \times G$$

where $G = \frac{R_4}{R_3} \times \left(1 + \frac{2 \times R_5}{R_{10}}\right)$

2. Select the feedback loop resistors R_5 and R_6:

Choose $R_5 = R_6 = 10\ k\Omega$ (Standard Value)

3. Select R_1, R_2, R_3, R_4. To set the Vref gain at 1V/V and avoid degrading the instrumentation amplifier's CMRR, ratios of R_4/R_3 and R_2/R_1 must be equal.

Choose $R_1 = R_2 = R_3 = R_4 = 10\ k\Omega$ (Standard Value)

4. Calculate R_{10} to meet the desired gain:

$$G = \frac{R_4}{R_3} \times \left(1 + \frac{2 \times R_5}{R_{10}}\right) = 10\ \frac{V}{V} \qquad (\quad)$$

$$R_4 = R_3 = 10\ k\Omega$$

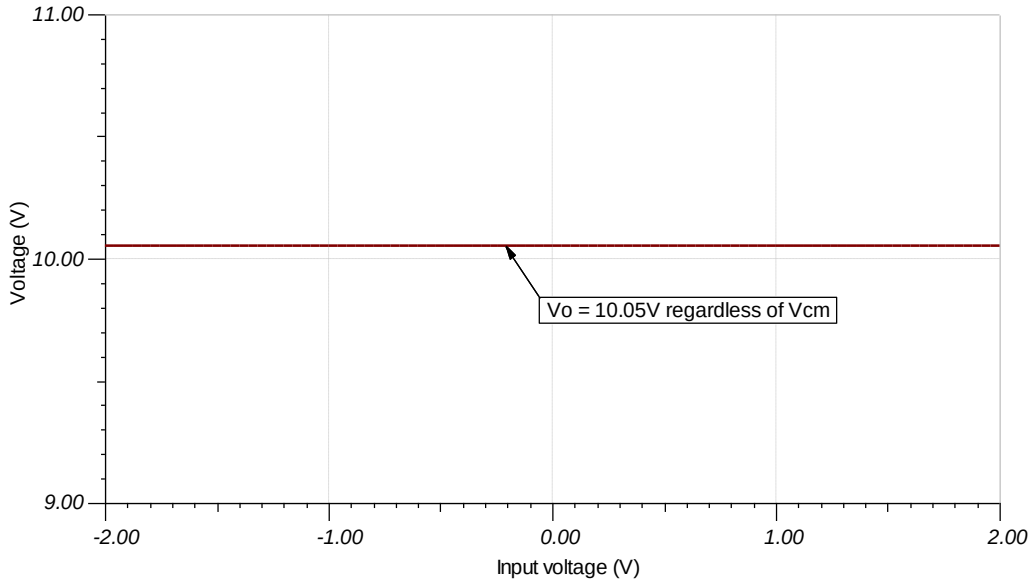
$$\rightarrow G = \left(1 + \frac{2 \times 10\ k\Omega}{R_{10}}\right) = 10\frac{V}{V} \rightarrow \left(1 + \frac{20k\Omega}{R_{10}}\right) = 10\frac{V}{V}$$

$$\frac{20\ k\Omega}{R_{10}} = 9\frac{V}{V} \rightarrow R_{10} = \frac{20k\Omega}{9} = 2222.2\ \Omega \rightarrow R_{10} = 2.21\ k\Omega \text{ (Standard Value)} \qquad (1)$$

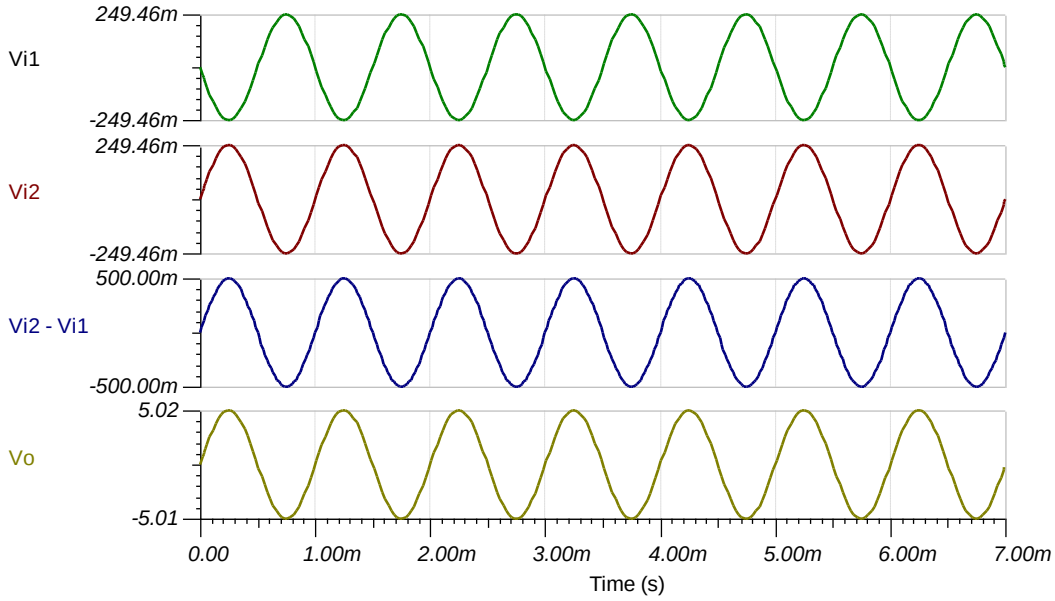
5. To check the common-mode voltage range, download and install the program from reference [5]. Edit the INA_Data.txt file in the installation directory by adding the code for a 3 op amp INA whose internal amplifiers have the common-mode range, output swing, and supply voltage range as defined by the amplifier of choice (TLV172 in this case). There is no V_{be} shift in this design and the gain of the output stage difference amplifeir is 1 V/V. The default supply voltage and reference voltages are ±15 V and 0 V, respectively. Run the program and set the gain and reference voltage accordingly. The resulting V_{CM} vs. V_{OUT} plot approximates the linear operating region of the discrete INA.

Design Simulations

DC Simulation Results



Transient Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAU8
3. TI Precision Labs
4. Instrumentation Amplifier V_{CM} vs. V_{OUT} Plots
5. Common-mode Range Calculator for Instrumentation Amplifiers

Design Featured Op Amp

TLV171	
V_{ss}	4.5V to 36V
V_{inCM}	(V–) – 0.1V < Vin < (V+) – 2V
V_{out}	Rail–to–rail
V_{os}	0.25mV
I_q	475µA
I_b	8pA
UGBW	3MHz
SR	1.5V/µs
#Channels	1,2,4
www.ti.com/product/tlv171	

Design Alternate Op Amp

	OPA172	OPA192
V_{ss}	4.5V to 36V	4.5V to 36V
V_{inCM}	(V–) – 0.1V < Vin < (V+) – 2V	V_{ee}–0.1V to V_{cc}+0.1V
V_{out}	Rail–to–rail	Rail–to–rail
V_{os}	0.2mV	±5µV
I_q	1.6mA	1mA/Ch
I_b	8pA	5pA
UGBW	10MHz	10MHz
SR	10V/µs	20V/µs
#Channels	1,2,4	1, 2, 4
	www.ti.com/product/opa172	www.ti.com/product/opa192

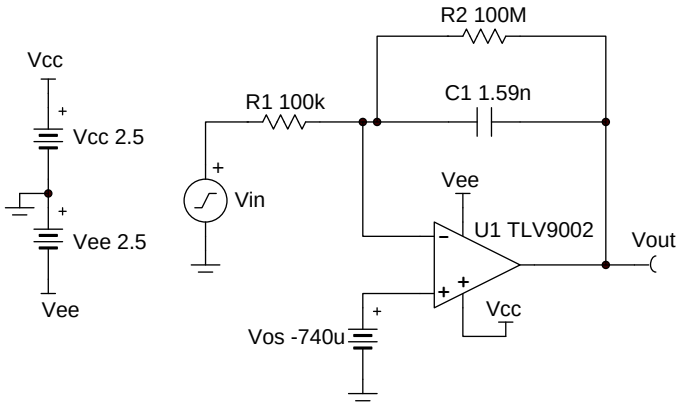
Integrator circuit

Design Goals

Input			Output		Supply	
f_{Min}	f_{0dB}	f_{Max}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}
100Hz	1kHz	100kHz	–2.45V	2.45V	2.5V	–2.5V

Design Description

The integrator circuit outputs the integral of the input signal over a frequency range based on the circuit time constant and the bandwidth of the amplifier. The input signal is applied to the inverting input so the output is inverted relative to the polarity of the input signal. The ideal integrator circuit will saturate to the supply rails depending on the polarity of the input offset voltage and requires the addition of a feedback resistor, R_2, to provide a stable DC operating point. The feedback resistor limits the lower frequency range over which the integration function is performed. This circuit is most commonly used as part of a larger feedback/servo loop which provides the DC feedback path, thus removing the requirement for a feedback resistor.



Design Notes

1. Use as large of a value as practical for the feedback resistor.
2. Select a CMOS op amp to minimize the errors from the input bias current.
3. The gain bandwidth product (GBP) of the amplifier will set the upper frequency range of the integrator function. The effectiveness of the integration function is usually reduced starting about one decade away from the amplifier bandwidth.
4. An adjustable reference needs to be connected to the non-inverting input of the op amp to cancel the input offset voltage or the large DC noise gain will cause the circuit to saturate. Op amps with very low offset voltage may not require this.

Design Steps

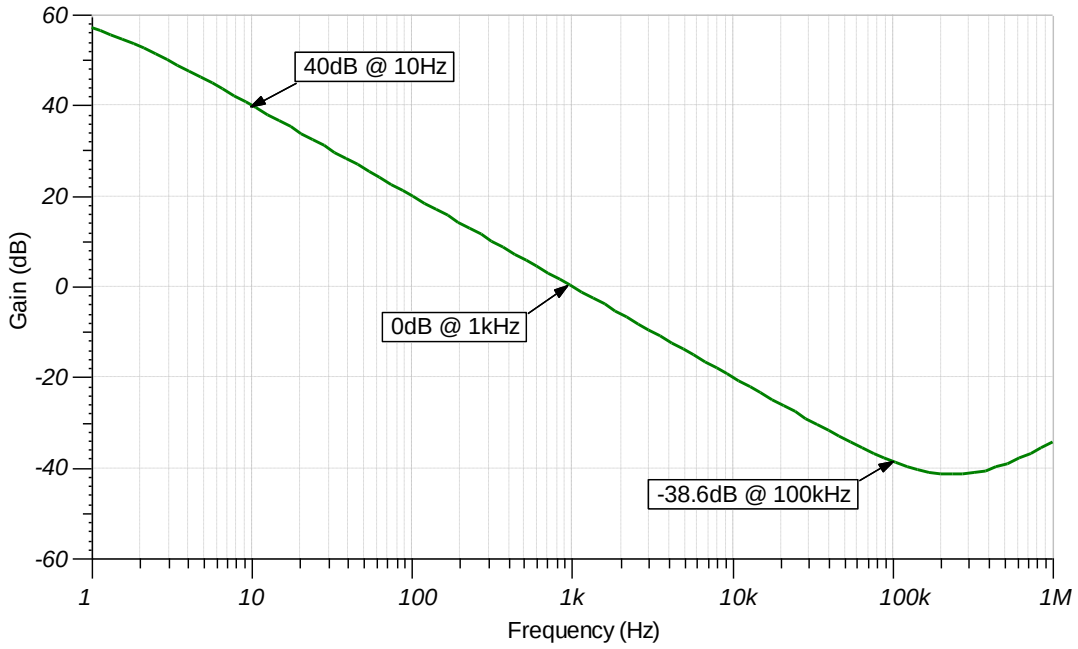
The ideal circuit transfer function is given below.

$$V_{out} = -\frac{1}{R_1 \times C_1}\int_0^t V_{in}(t)\,dt$$

1. Set R_1 to a standard value.
   $R_1 = 100k\Omega$
2. Calculate C_1 to set the unity-gain integration frequency.
   $$C_1 = \frac{1}{2 \times \pi \times R_1 \times f_{0dB}} = \frac{1}{2 \times \pi \times 100k\Omega \times 1\ kHz} = 1.59nF$$
3. Calculate R_2 to set the lower cutoff frequency a decade less than the minimum operating frequency.
   $$R_2 \geq \frac{10}{2 \times \pi \times C_1 \times f_{Min}} \geq \frac{10}{2 \times \pi \times 1.59nF \times 10Hz} \geq 100M\Omega$$
4. Select an amplifier with a gain bandwidth at least 10 times the desired maximum operating frequency.
   $GBP \geq 10 \times f_{Max} \geq 10 \times 100kHz \geq 1\ \ MHz$

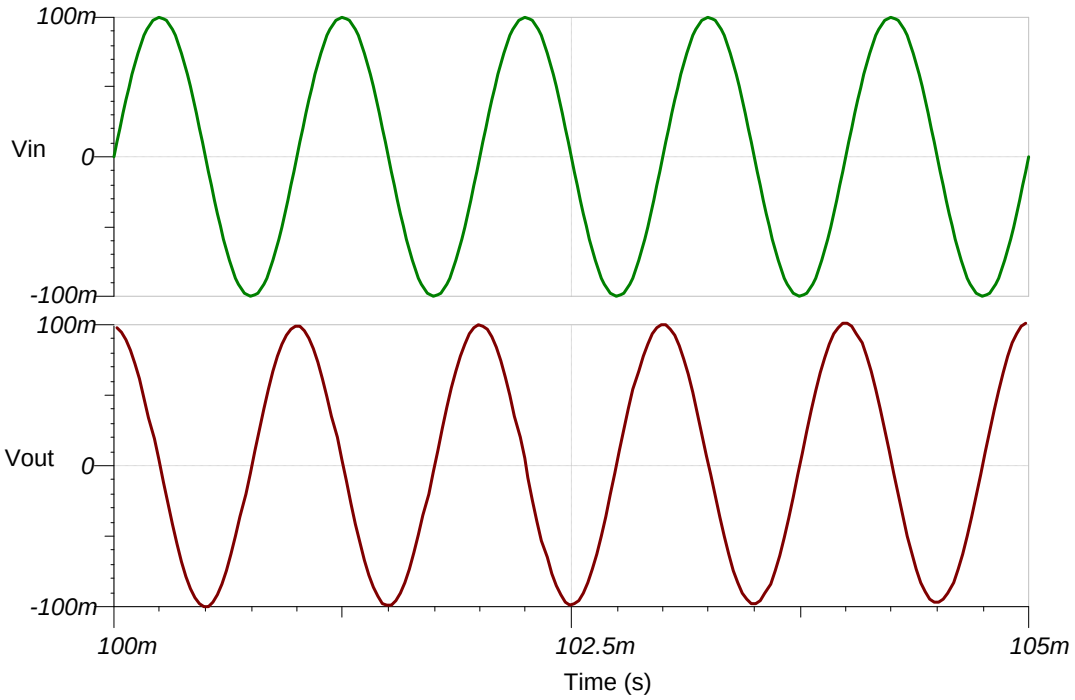
Design Simulations

AC Simulation Results

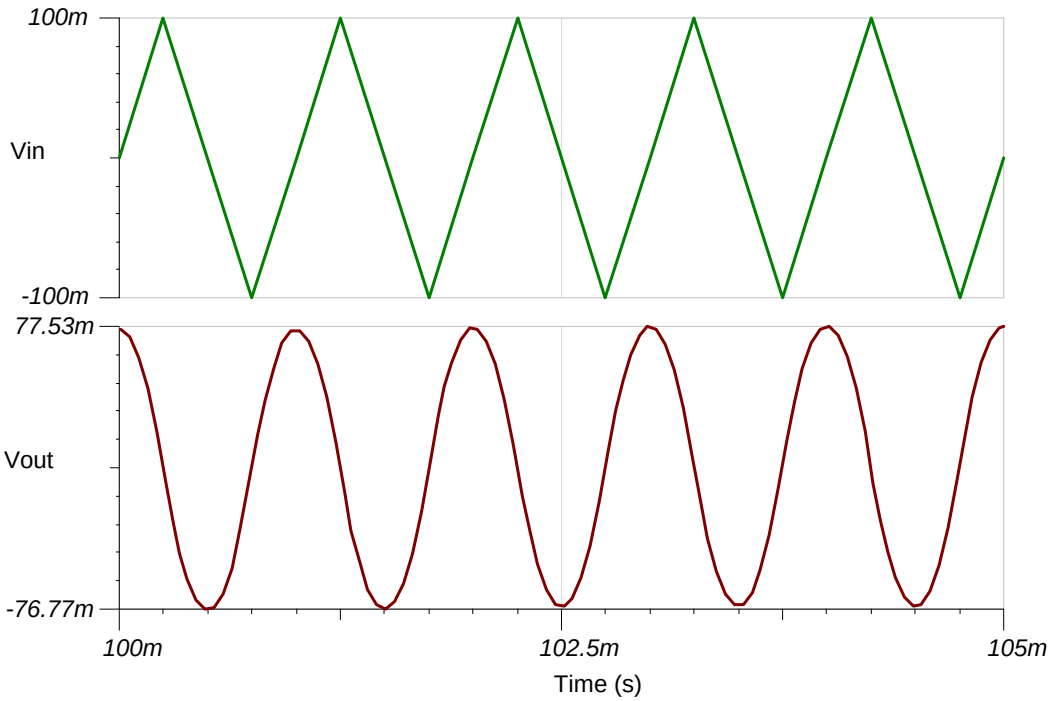


Transient Simulation Results

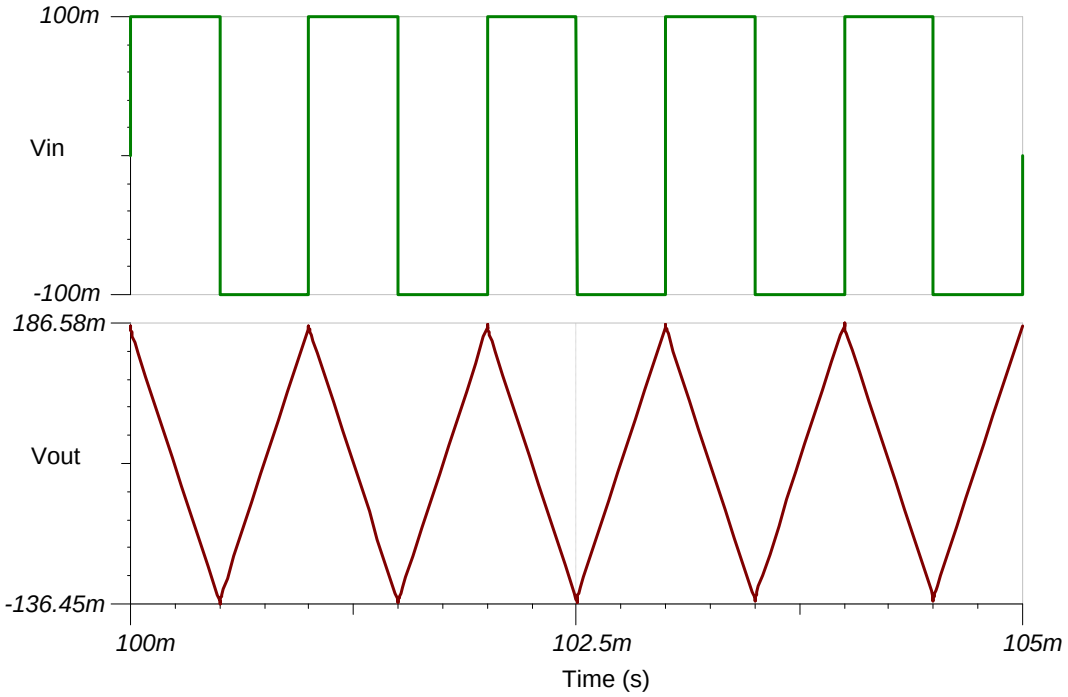
A 1-kHz sine wave input yields a 1-kHz cosine output.



A 1-kHz triangle wave input yields a 1-kHz sine wave output.



A 1-kHz square wave input yields a 1-kHz triangle wave output.



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC496.

See TIPD191, www.ti.com/tool/tipd191.

Design Featured Op Amp

TLV9002	
V_{cc}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.4mV
I_q	0.06mA
I_b	5pA
UGBW	1MHz
SR	2V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv9002	

Design Alternate Op Amp

OPA376	
V_{cc}	2.2V to 5.5V
V_{inCM}	(V_{ee}-0.1V) to (V_{cc}-1.3V)
V_{out}	Rail-to-rail
V_{os}	0.005mV
I_q	0.76mA
I_b	0.2pA
UGBW	5.5MHz
SR	2V/µs
#Channels	1, 2, 4
www.ti.com/product/opa376	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

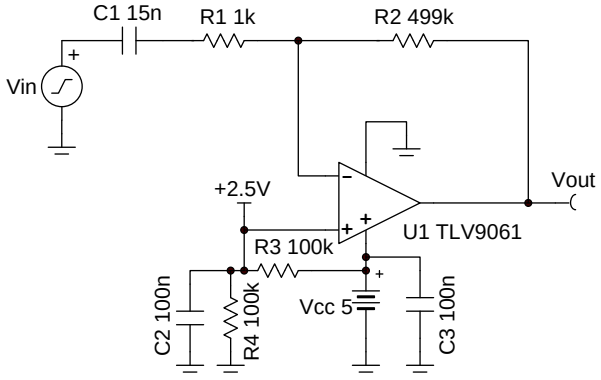
Differentiator circuit

Design Goals

Input		Output		Supply		
f_{Min}	f_{Max}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
100Hz	5kHz	0.1V	4.9V	5V	0V	2.5V

Design Description

The differentiator circuit outputs the derivative of the input signal over a frequency range based on the circuit time constant and the bandwidth of the amplifier. The input signal is applied to the inverting input so the output is inverted relative to the polarity of the input signal. The ideal differentiator circuit is fundamentally unstable and requires the addition of an input resistor, a feedback capacitor, or both, to be stable. The components required for stability limit the bandwidth over which the differentiator function is performed.



Design Notes

1. Select a large resistance for R_2 to keep the value of C_1 reasonable.
2. A capacitor can be added in parallel with R_2 to filter the high-frequency noise of the circuit. The capacitor will limit the effectiveness of the differentiator function starting about half a decade (approximately 3.5 times) away from the filter cutoff frequency.
3. A reference voltage can be applied to the non-inverting input to set the DC output voltage which allows the circuit to work single-supply. The reference voltage can be derived from a voltage divider.
4. Operate within the linear output voltage swing (see Aol specification) to minimize non-linearity errors.

Design Steps

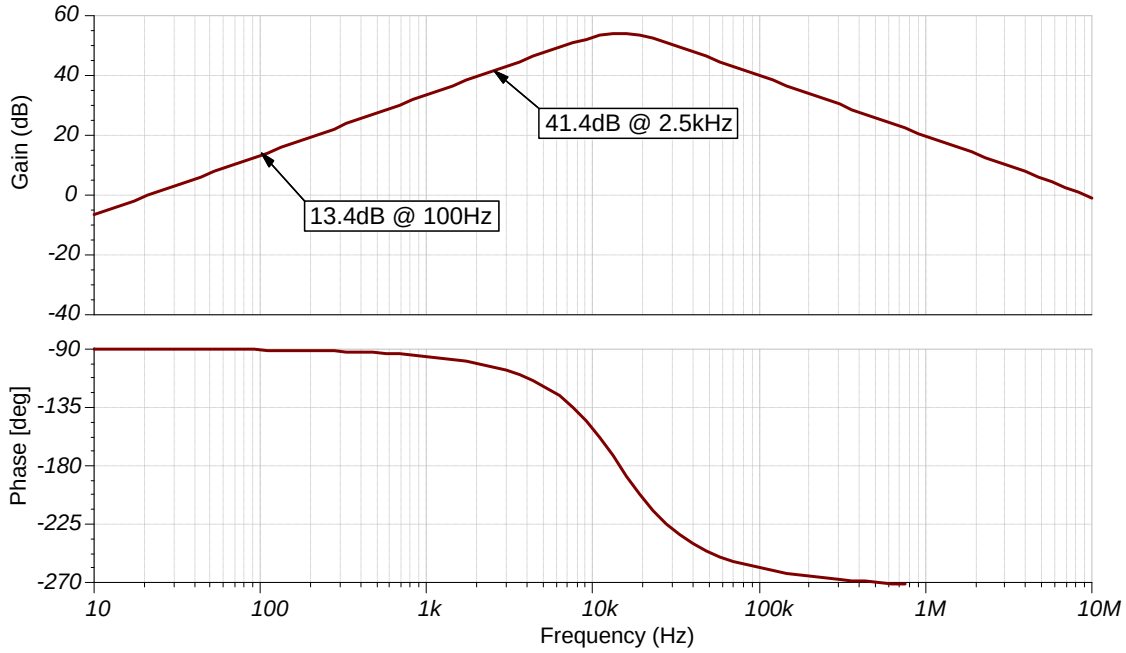
The ideal circuit transfer function is given below.

$$Vout = -R_2 \times C_1 \times \frac{dV_{in}(t)}{dt}$$

1. Set R_2 to a large standard value.
   $R_2 = 499k\Omega$
2. Set the minimum differentiation frequency at least half a decade below the minimum operating frequency.
   $C_1 \geq \frac{3.5}{2 \times \pi \times R_2 \times f_{min}} \geq \frac{3.5}{2 \times \pi \times 499k\Omega \times 100Hz} \geq 11.1\ nF \approx 15nF\ (Standard\ Value)$
3. Set the upper cutoff frequency at least half a decade above the maximum operating frequency.
   $R_1 \leq \frac{1}{3.5 \times 2 \times \pi \times C_1 \times f_{Max}} \leq \frac{1}{7 \times \pi \times 15nF \times 2.5kHz} \leq 1.2k\Omega \approx 1\ k\Omega\ (Standard\ Value)$
4. Calculate the necessary op amp gain bandwidth product (GBP) for the circuit to be stable.
   $GBP > \frac{R_1 + R_2}{2 \times \pi \times R_1^2 \times C_1} > \frac{499k\Omega + 1\ k\Omega}{2 \times \pi \times 1\ k\Omega^2 \times 15nF} > 5.3MHz$
   - The bandwidth of the TLV9061 is 10MHz, therefore this requirement is met.
5. If a feedback capacitor, C_F , is added in parallel with R_2, the equation to calculate the cutoff frequency follows.
   $f_c = \frac{1}{2 \times \pi \times R_2 \times C_F}$
6. Calculate the resistor divider values for a 2.5-V reference voltage.
   $R_3 = \frac{V_{cc} - V_{ref}}{V_{ref}} \times R_4 = \frac{5V - 2.5V}{2.5V} \times R_4 = R_4$
   $R_3 = R_4 = 100k\Omega\ (Standard\ Values)$

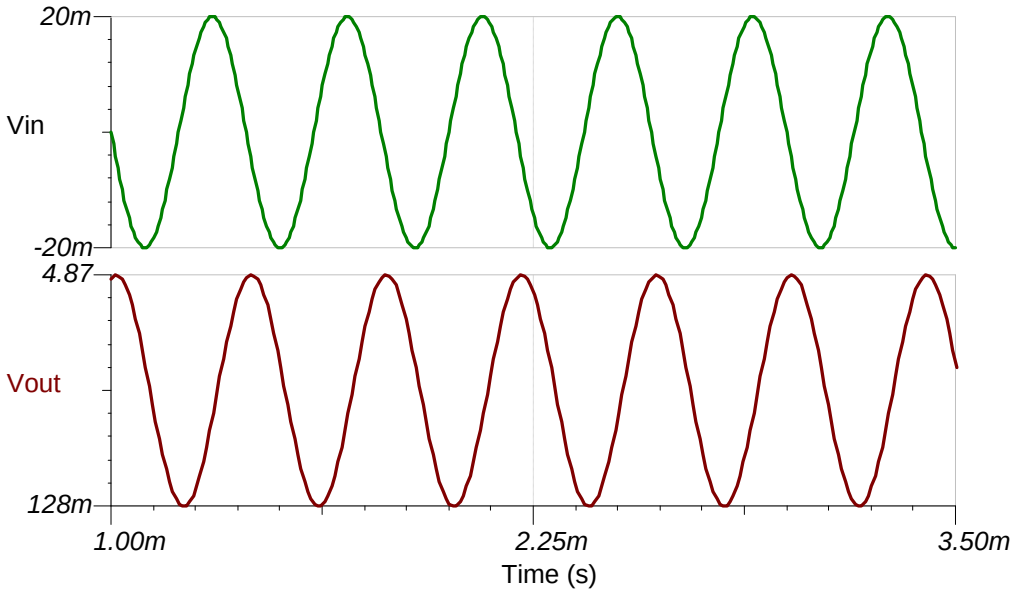
Design Simulations

AC Simulation Results

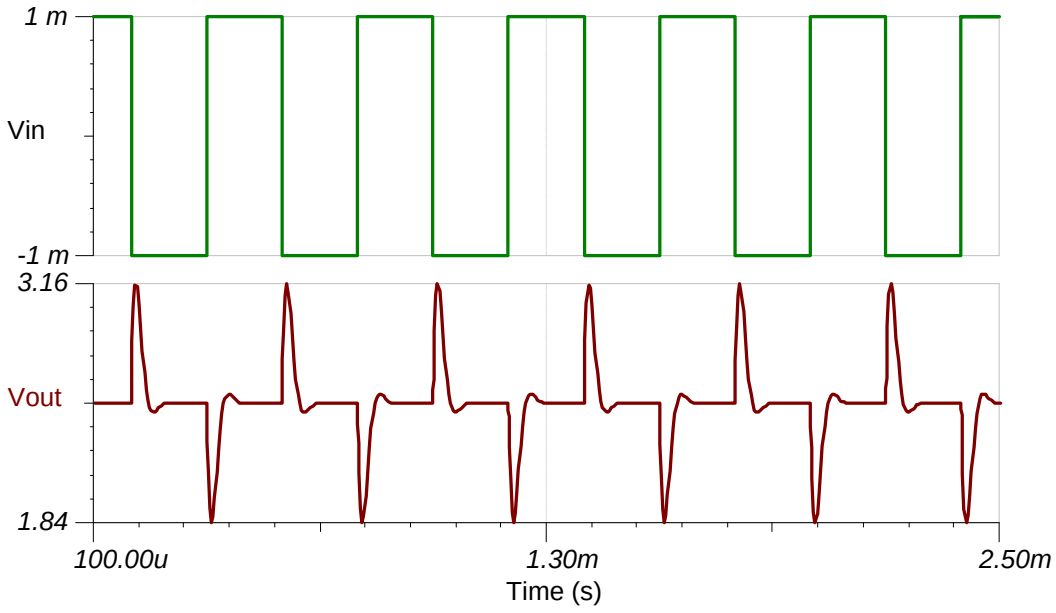


Transient Simulation Results

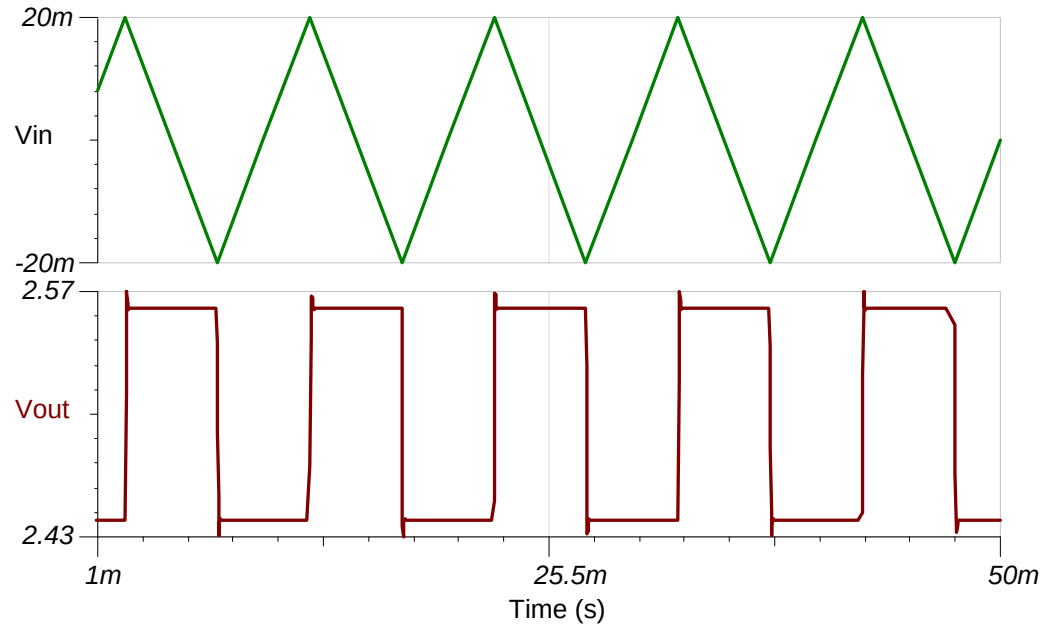
A 2.5-kHz sine wave input yields a 2.5-kHz cosine output.



A 2.5-kHz square wave input produces an impulse output.



A 100-Hz triangle wave input yields a square wave output.



Design Featured Op Amp

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC497.

TLV9061	
V_{cc}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.3mV
I_q	0.538mA
I_b	0.5pA
UGBW	10MHz
SR	6.5V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv9061	

Design Alternate Op Amp

OPA374	
V_{cc}	2.3V to 5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1mV
I_q	0.585mA
I_b	0.5pA
UGBW	6.5MHz
SR	0.4V/µs
#Channels	1, 2, 4
www.ti.com/product/opa374	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

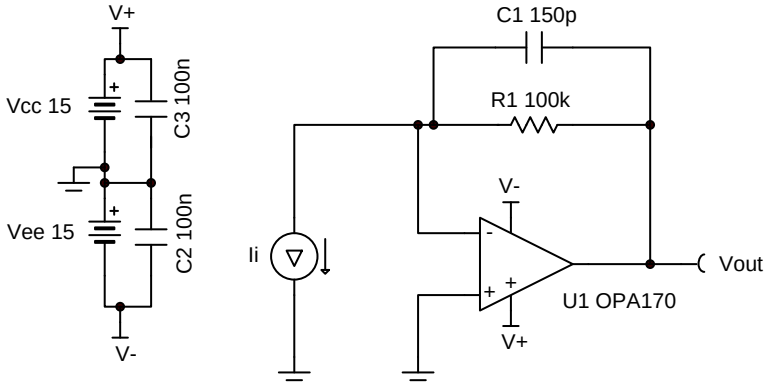
Transimpedance amplifier circuit

Design Goals

Input		Output		BW	Supply	
I_{iMin}	I_{iMax}	V_{oMin}	V_{oMax}	f_p	V_{cc}	V_{ee}
0A	50µA	0V	5V	10kHz	15V	–15V

Design Description

The transimpedance op amp circuit configuration converts an input current source into an output voltage. The current to voltage gain is based on the feedback resistance. The circuit is able to maintain a constant voltage bias across the input source as the input current changes which benefits many sensors.



Design Notes

1. Use a JFET or CMOS input op amp with low bias current to reduce DC errors.
2. A bias voltage can be added to the non-inverting input to set the output voltage for 0-A input currents.
3. Operate within the linear output voltage swing (see A_{ol} specification) to minimize non-linearity errors.

Design Steps

1. Select the gain resistor.

$$R_1 = \frac{V_{oMax} - V_{oMin}}{I_{iMax}} = \frac{5V - 0V}{50\mu A} = 100k\Omega$$

2. Select the feedback capacitor to meet the circuit bandwidth.

$$C_1 \leq \frac{1}{2 \times \pi \times R_1 \times f_p}$$

$$C_1 \leq \frac{1}{2 \times \pi \times 100k\Omega \times 10kHz} \leq 159pF \approx 150pF \text{ (Standard Value)}$$

3. Calculate the necessary op amp gain bandwidth (GBW) for the circuit to be stable.

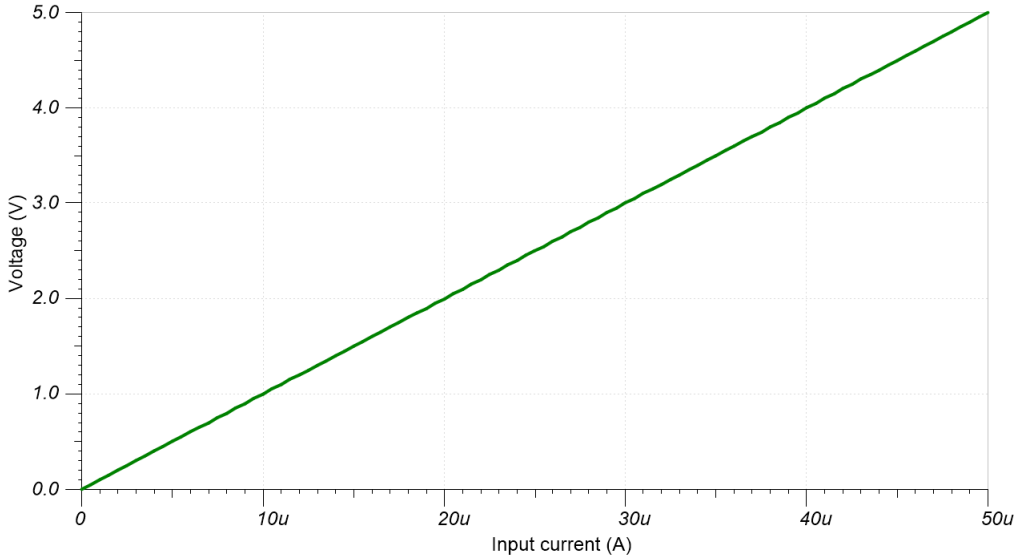
$$GBW > \frac{C_i + C_1}{2 \times \pi \times R_1 \times C_1^2} > \frac{6pF + 150pF}{2 \times \pi \times 100k\Omega \times (150pF)^2} > 11.03kHz$$

where $C_i = C_s + C_d + C_{cm} = 0pF + 3pF + 3pF = 6pF$ given

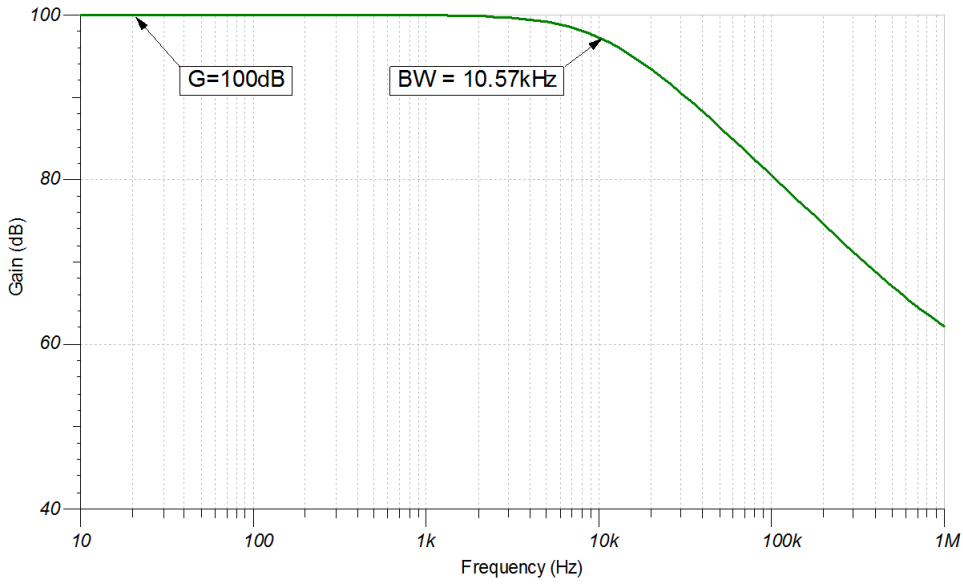
- C_s: Input source capacitance
- C_d: Differential input capacitance of the amplifier
- C_{cm}: Common-mode input capacitance of the inverting input

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC501.

See TIPD176, www.ti.com/tool/tipd176.

Design Featured Op Amp

OPA170	
V_{cc}	2.7V to 36V
V_{inCM}	(V_{ee}–0.1V) to (V_{cc} –2V)
V_{out}	Rail–to–rail
V_{os}	0.25mV
I_q	0.11mA
I_b	8pA
UGBW	1.2MHz
SR	0.4V/µs
#Channels	1, 2, 4
www.ti.com/product/opa170	

Design Alternate Op Amp

OPA1671	
V_{cc}	1.7V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	(V_{ee}+10mV) to (V_{cc}-10mV) @ 275µA
V_{os}	250µV
I_q	940µA
I_b	1pA
UGBW	12MHz
SR	5V/µs
#Channels	1
www.ti.com/product/opa1671	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Updated Design Alternate Op Amp table with OPA1671. Added link to circuit cookbook landing page.

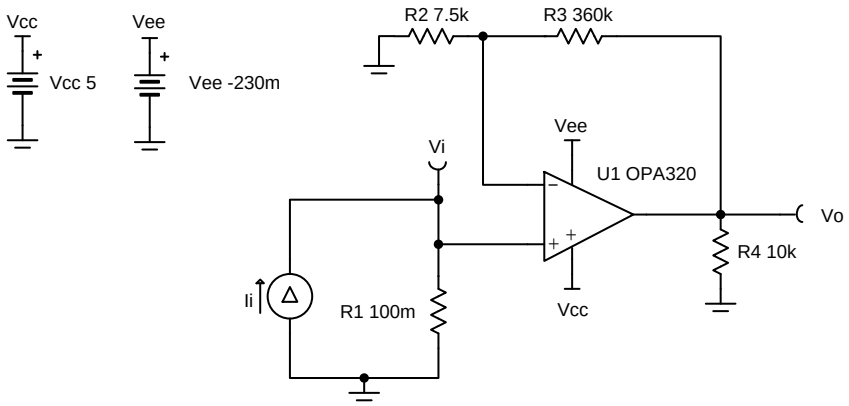
Single-supply, low-side, unidirectional current-sensing solution with output swing to GND circuit

Design Goals

Input		Output		Supply		
I_{iMin}	I_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
0A	1A	0V	4.9V	5V	0V	0V

Design Description

This single-supply, low-side, current sensing solution accurately detects load current between 0A to 1A and converts it to a voltage between 0V to 4.9V. The input current range and output voltage range can be scaled as necessary and larger supplies can be used to accommodate larger swings. A negative charge pump (such as the LM7705) is used as the negative supply in this design to maintain linearity for output signals near 0V.



Design Notes

1. Use precision resistors to minimize gain error.
2. For light load accuracy, the negative supply should extend slightly below ground.
3. A capacitor placed in parallel with the feedback resistor will limit bandwidth and help reduce noise.

Design Steps

1. Determine the transfer function.

   $V_o = I_i \times R_1 \times \left(1 + \frac{R_3}{R_2}\right)$

2. Define the full-scale shunt voltage and shunt resistance.

   $V_{iMax} = 100mV$ at $I_{iMax} = 1A$

   $R_1 = \frac{V_{iMax}}{I_{iMax}} = \frac{100mV}{1\ A} = 100m\Omega$

3. Select gain resistors to set the output range.

   $V_{iMax} = 100mV$ and $V_{oMax} = 4.9V$

   $Gain = \frac{V_{oMax}}{V_{iMax}} = \frac{4.9V}{100mV} = 49\frac{V}{V}$

   $Gain = 1 + \frac{R_3}{R_2} = 49\frac{V}{V}$

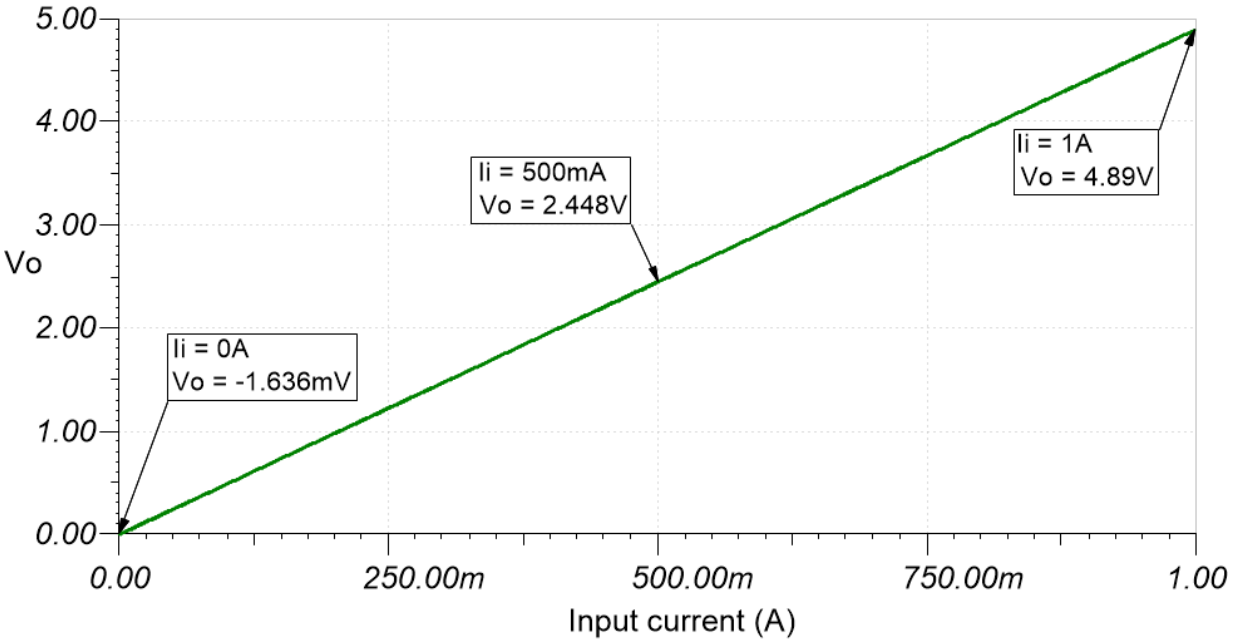
4. Select a standard value for R_2 and R_3.

   $R_2 = 7.5k\Omega$ (0.05% Standard Value)

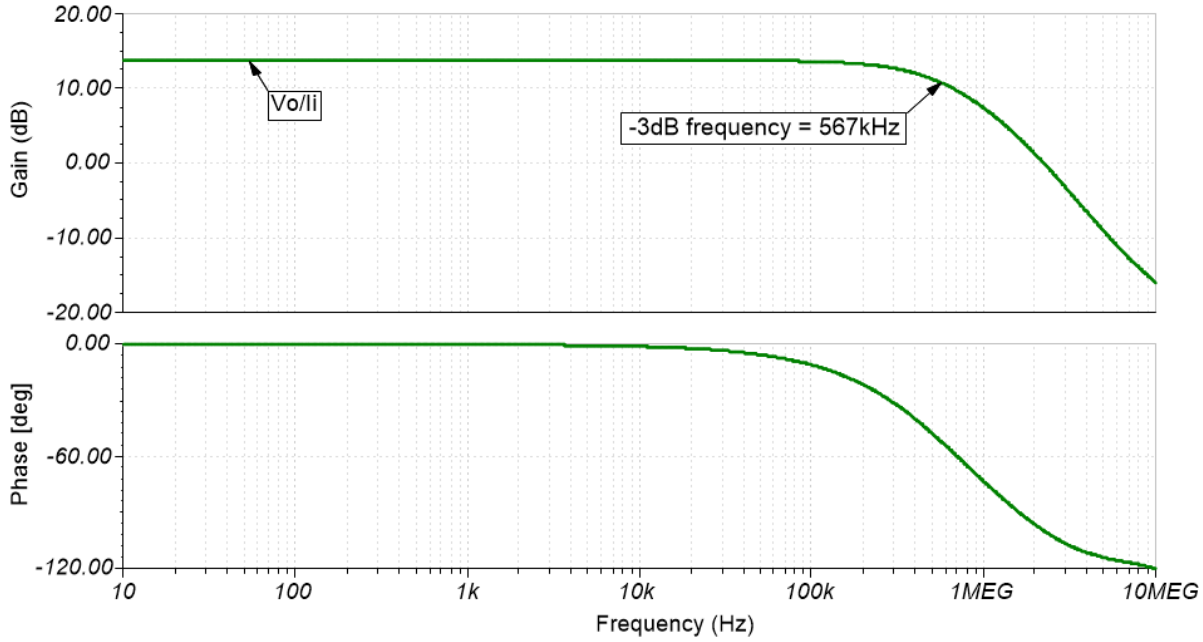
   $R_3 = 48 \times R_2 = 360k\Omega$ (0.05% Standard Value)

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC499.

See TIPD129, www.ti.com/tool/tipd129.

Design Featured Op Amp

OPA320	
V_{cc}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	40µV
I_q	1.5mA/Ch
I_b	0.2pA
UGBW	10MHz
SR	10V/µs
#Channels	1, 2
www.ti.com/product/opa320	

Design Alternate Op Amp

TLV9002	
V_{cc}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	400µV
I_q	60µA
I_b	5pA
UGBW	1MHz
SR	2V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv9002	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

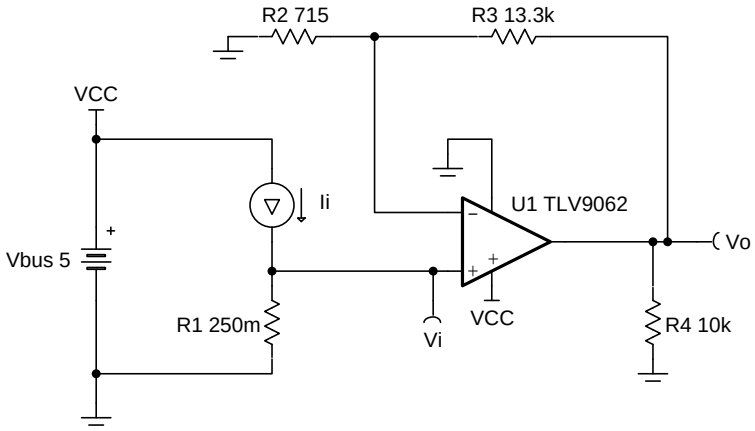
Single-supply, low-side, unidirectional current-sensing circuit

Design Goals

Input		Output		Supply		Full–Scale Range Error
I_{iMax}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	FSR_{Error}
1A	250mV	50mV	4.9V	5V	0V	0.2%

Design Description

This single–supply, low–side, current sensing solution accurately detects load current up to 1A and converts it to a voltage between 50mV and 4.9V. The input current range and output voltage range can be scaled as necessary and larger supplies can be used to accommodate larger swings.



Design Notes

1. Use the op amp linear output operating range, which is usually specified under the test conditions.
2. The common–mode voltage is equal to the input voltage.
3. Tolerance of the shunt resistor and feedback resistors will determine the gain error of the circuit.
4. Avoid placing capacitive loads directly on the output of the amplifier to minimize stability issues.
5. If trying to detect zero current with output swing to GND, a negative charge pump (such as LM7705) can be used as the negative supply in this design to maintain linearity for output signals near 0V. [5]
6. Using high–value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
7. The small–signal bandwidth of this circuit depends on the gain of the circuit and gain bandwidth product (GBP) of the amplifier.
8. Filtering can be accomplished by adding a capacitor in parallel with R_3 . Adding a capacitor in parallel with R_3 will also improve stability of the circuit if high–value resistors are used.
9. For more information on op amp linear operating region, stability, capacitive load drive, driving ADCs, and bandwidth please see the Design References section.

Design Steps

The transfer function for this circuit is given below.

$$V_o = I_i \times R_1 \times \left(1 + \frac{R_3}{R_2}\right)$$

1. Define the full–scale shunt voltage and calculate the maximum shunt resistance.

   $V_{iMax} = 250\ mV \quad at \quad I_{iMax} = 1\ A$

   $$R_1 = \frac{V_{iMax}}{I_{iMax}} = \frac{250\ mV}{1\ A} = 250\ m\Omega$$

2. Calculate the gain required for maximum linear output voltage.

   $V_{iMax} = 250\ mV \quad and \quad V_{oMax} = 4.9\ V$

   $$Gain = \frac{V_{oMax}}{V_{iMin}} = \frac{4.9\ V}{250\ mV} = 19.6\ \frac{V}{V}$$

3. Select standard values for R_2 and R_3.

   From Analog Engineer's calculator, use "Find Amplifier Gain" and get resistor values by inputting gain ratio of 19.6.

   R_2 = 715 Ω (0.1% Standard Value)

   R_3 = 13.3 kΩ (0.1% Standard Value)

4. Calculate minimum input current before hitting output swing–to–rail limit. I_{iMin} represents the minimum accurately detectable input current.

   $V_{oMin} = 50\ mV; \quad R_1 = 250\ m\Omega$

   $$V_{iMin} = \frac{V_{oMin}}{Gain} = \frac{50\ mV}{19.6\ \frac{V}{V}} = 2.55\ mV$$

   $$I_{iMin} = \frac{V_{iMin}}{R_1} = \frac{2.55\ mV}{250\ m\Omega} = 10.2\ mA$$

5. Calculate Full scale range error and relative error. V_{os} is the typical offset voltage found in datasheet.

   $$FSR_{error} = \left(\frac{V_{os}}{V_{iMax} - V_{iMin}}\right) \times 100 = \left(\frac{0.3\ mV}{247.45\ mV}\right) \times 100 = 0.121\ \%$$

   $$\text{Relative Error at } I_{iMax} = \left(\frac{V_{os}}{V_{iMax}}\right) \times 100 = \left(\frac{0.3\ mV}{250\ mV}\right) \times 100 = 0.12\ \%$$

   $$\text{Relative Error at } I_{iMin} = \left(\frac{V_{os}}{V_{iMin}}\right) \times 100 = \left(\frac{0.3\ mV}{2.5\ mV}\right) \times 100 = 12\ \%$$

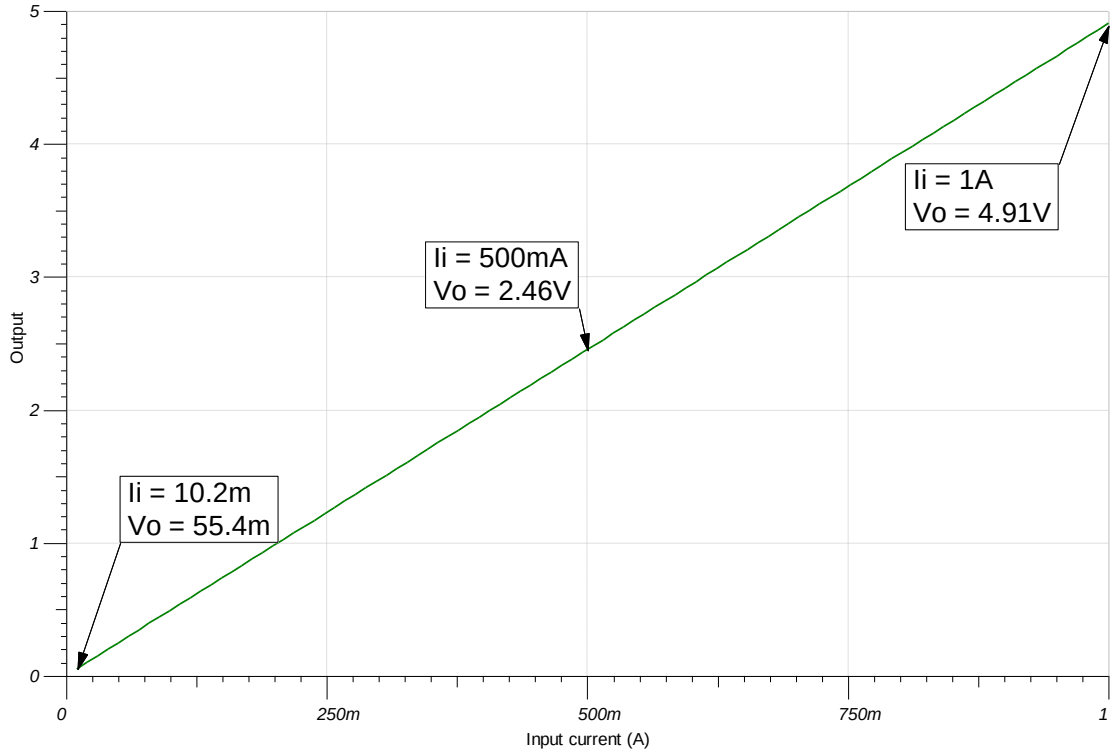
6. To maintain sufficient phase margin, ensure that the zero created by the gain setting resistors and input capacitance of the device is greater than the bandwidth of the circuit

   $$\frac{1}{2 \times \pi \times (C_{cm} + C_{diff}) \times (R_2 \| R_3)} > \frac{GBP}{G}$$

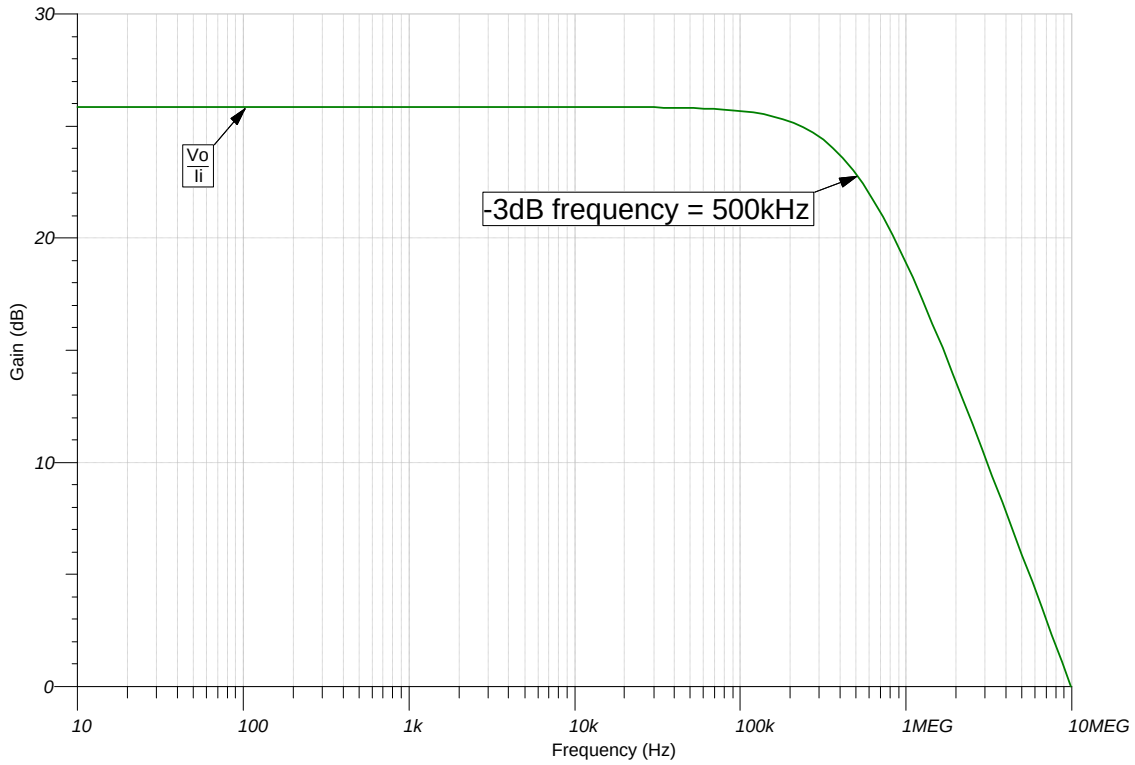
   $$\frac{1}{2 \times \pi \times (3pF + 3pF) \times \left(\frac{715\ \Omega \times 13.3\ k\Omega}{715\ \Omega + 13.3\ k\Omega}\right)} > \frac{10\ MHz}{19.6\ \frac{V}{V}} = 39.1\ MHz > 510\ kHz$$

Design Simulations

DC Simulation Results



AC Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOC523
3. TI Precision Designs TIPD129, TIPD104
4. TI Precision Labs
5. Single-Supply, Low-Side, Unidirectional Current-Sensing Solution with Output Swing to GND Circuit

Design Featured Op Amp

TLV9061	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	0.3mV
I_q	538µA
I_b	0.5pA
UGBW	10MHz
SR	6.5V/µs
#Channels	1,2,4
www.ti.com/product/tlv9061	

Design Alternate Op Amp

OPA375	
V_{cc}	2.25V to 5.5V
V_{inCM}	(V–) to ((V+)–1.2V)
V_{out}	Rail–to–rail
V_{os}	0.15mV
I_q	890µA
I_b	10pA
UGBW	10MHz
SR	4.75V/µs
#Channels	1
www.ti.com/product/OPA375	

For battery operated or power conscious designs, outside of the original design goals described earlier, where lowering total system power is desired.

LPV821	
V_{cc}	1.7V to 3.6V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	1.5µV
I_q	650nA/Ch
I_b	7pA
UGBW	8kHz
SR	3.3V/ms
#Channels	1
www.ti.com/product/LPV821	

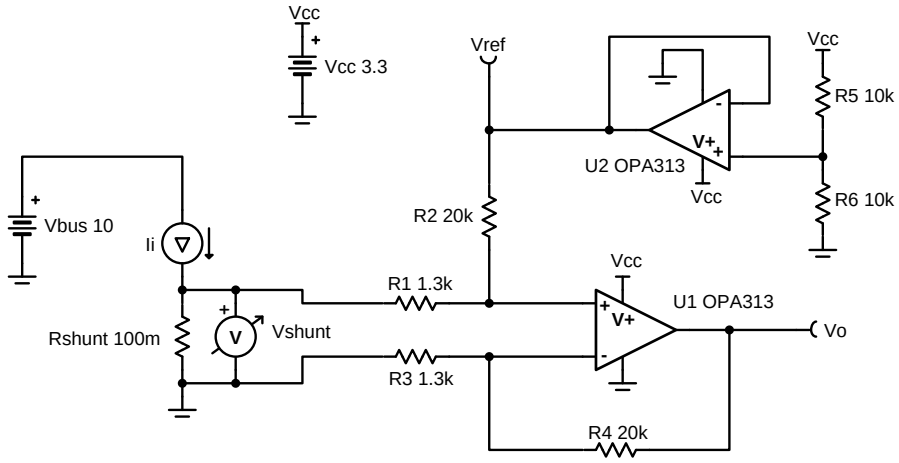
Low-side, bidirectional current sensing circuit

Design Goals

Input		Output		Supply		
I_{iMin}	I_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–1A	1A	110mV	3.19V	3.3V	0V	1.65V

Design Description

This single-supply low-side, bidirectional current sensing solution can accurately detect load currents from –1A to 1A. The linear range of the output is from 110mV to 3.19V. Low-side current sensing keeps the common-mode voltage near ground, and is thus most useful in applications with large bus voltages.



Design Notes

1. To minimize errors, set $R_3 = R_1$ and $R_4 = R_2$.
2. Use precision resistors for higher accuracy.
3. Set output range based on linear output swing (see A_{ol} specification).
4. Low-side sensing should not be used in applications where the system load cannot withstand small ground disturbances or in applications that need to detect load shorts.

Design Steps

1. Determine the transfer equation given $R_4 = R_2$ and $R_1 = R_3$.

   $$V_o = (I_i \times R_{shunt} \times \frac{R_4}{R_3}) + V_{ref}$$

   $$V_{ref} = V_{cc} \times (\frac{R_6}{R_5 + R_6})$$

2. Determine the maximum shunt resistance.

   $$R_{shunt} = \frac{V_{shunt}}{I_{imax}} = \frac{100mV}{1\ A} = 100m\Omega$$

3. Set reference voltage.

   a. Since the input current range is symmetric, the reference should be set to mid supply. Therefore, make R_5 and R_6 equal.

      $$R_5 = R_6 = 10k\Omega$$

4. Set the difference amplifier gain based on the op amp output swing. The op amp output can swing from 100mV to 3.2V, given a 3.3-V supply.

   $$Gain = \frac{V_{oMax} - V_{oMin}}{R_{shunt} \times (I_{iMax} - I_{iMin})} = \frac{3.2V - 100mV}{100m\Omega \times (1\ A - (-1\ A))} = 15.5\ \frac{V}{V}$$

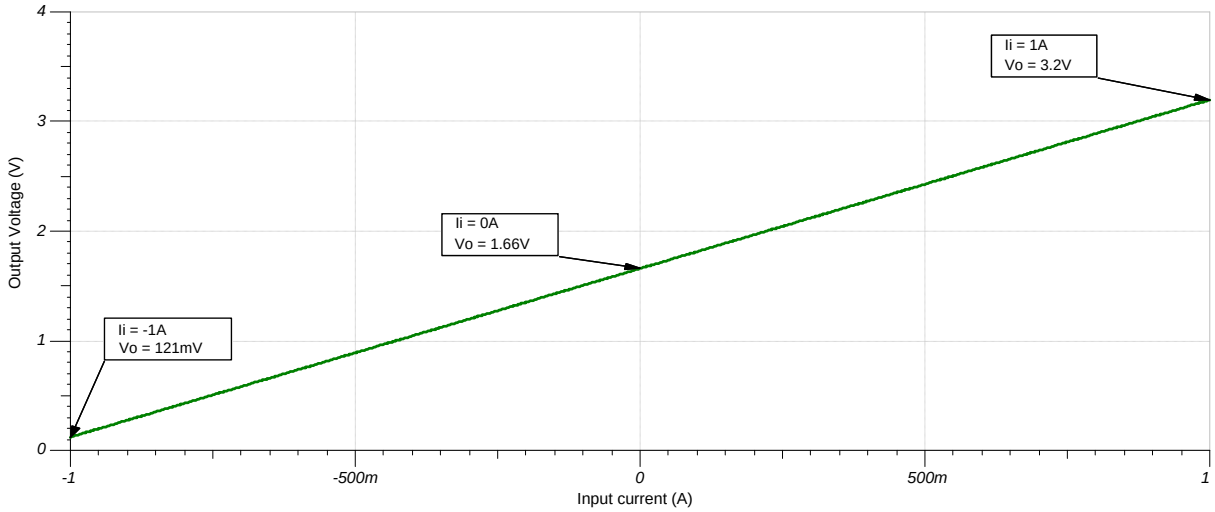
   $$Gain = \frac{R_4}{R_3} = 15.5\ \frac{V}{V}$$

   $$\text{Choose } R_1 = R_3 = 1.3k\Omega \text{ (Standard Value)}$$

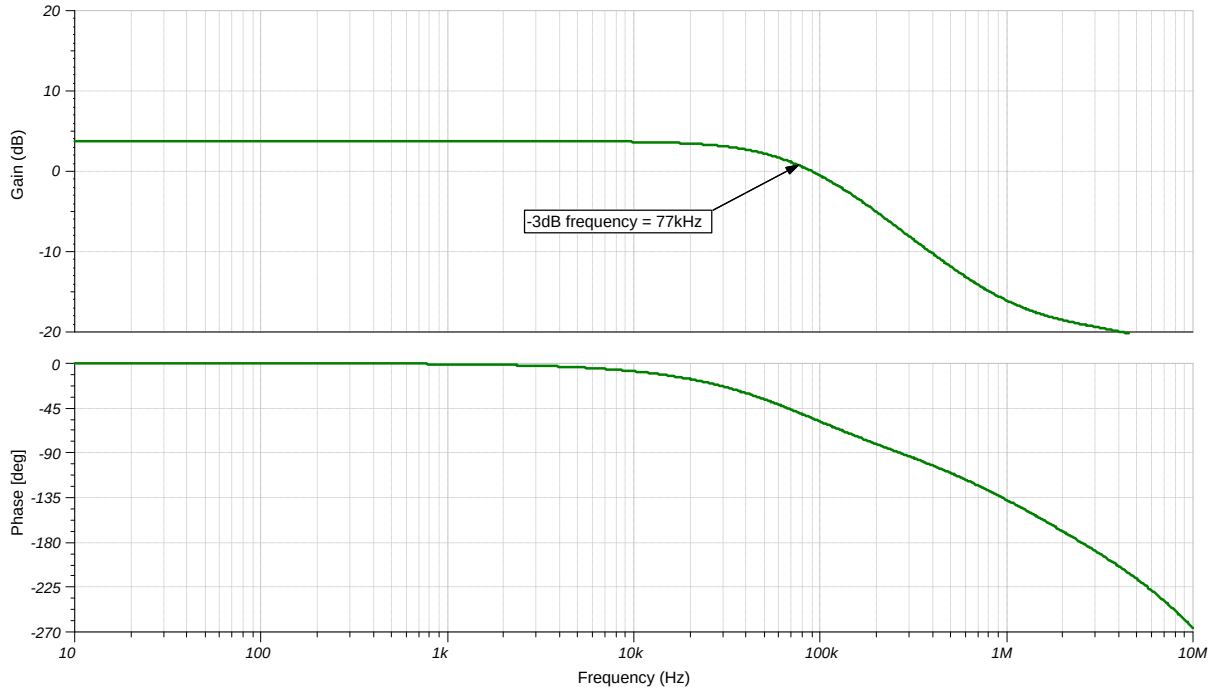
   $$R_2 = R_4 = 15.5\frac{V}{V} \times 1.3k\Omega = 20.15\ k\Omega \approx 20k\Omega \text{ (Standard Value)}$$

Design Simulations

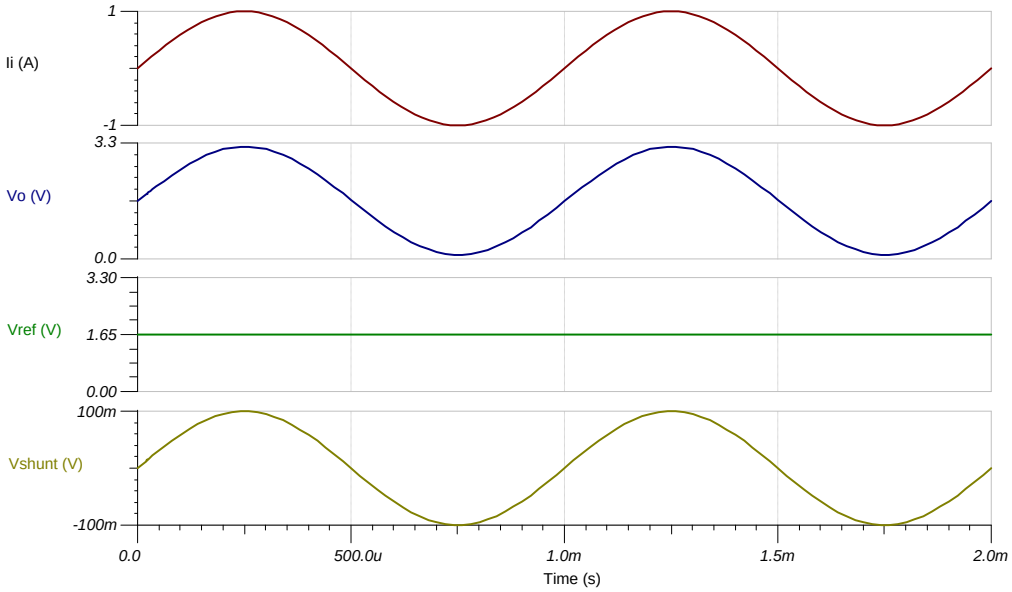
DC Simulation Results



Closed Loop AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC500.

See TIPD175, www.ti.com/tipd175.

Design Featured Op Amp

OPA313	
V_{cc}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	500µV
I_q	50µA/Ch
I_b	0.2pA
UGBW	1MHz
SR	0.5V/µs
#Channels	1, 2, 4
www.ti.com/product/opa313	

Design Alternate Op Amp

	TLV9062	OPA376
V_{cc}	1.8V to 5.5V	2.2V to 5.5V
V_{inCM}	Rail-to-rail	Rail-to-rail
V_{out}	Rail-to-rail	Rail-to-rail
V_{os}	300µV	5µV
I_q	538µA/Ch	760µA/Ch
I_b	0.5pA	0.2pA
UGBW	10MHz	5.5MHz
SR	6.5V/µs	2V/µs
#Channels	1, 2, 4	1, 2, 4
	www.ti.com/product/tlv9062	www.ti.com/product/opa376

For battery-operated or power-conscious designs, outside of the original design goals described earlier, where lowering total system power is desired.

LPV821	
V_{cc}	1.7V to 3.6V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1.5µV
I_q	650nA/Ch
I_b	7pA
UGBW	8KHz
SR	3.3V/ms
#Channels	1
www.ti.com/product/lpv821	

Revision History

Revision	Date	Change
B	January 2019	Downscale the title. Added link to circuit cookbook landing page.
A	May 2018	Changed title role to 'Amplifiers'. Added SPICE simulation file link. Added LPV821 as a Design Alternate Op Amp for battery-operated or power-conscious designs.

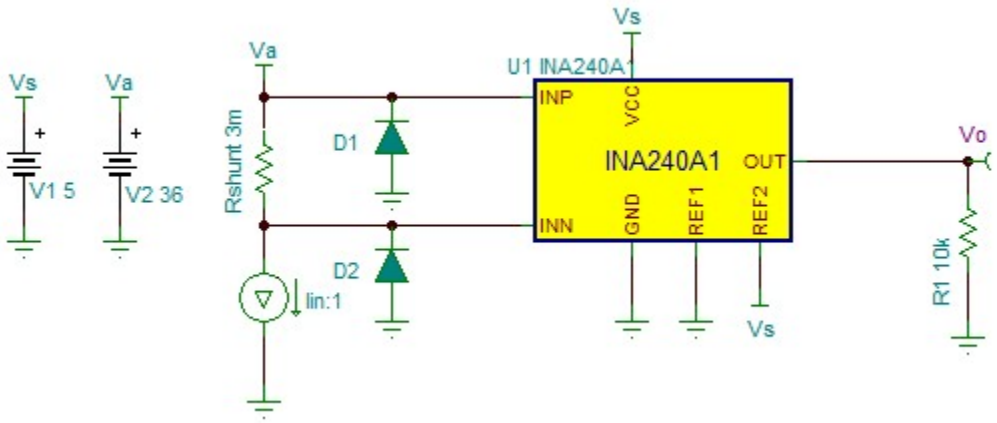
High-side, bidirectional current-sensing circuit with transient protection

Design Goals

Input		Output		Supply			Standoff and Clamp Voltages		EFT Level
I_{inMin}	I_{inMax}	V_{oMin}	V_{oMax}	Vs	GND	V_{ref}	Vwm	Vc	Vpp
–40A	40A	100mV	4.9V	5V	0V	2.5V	36V	80V	2kV 8/20µs

Design Description

This high-side, bidirectional current sensing solution can accurately measure current in the range of –40A to 40A for a 36-V voltage bus. The linear voltage output is 100mV to 4.90V. This solution is also designed to survive IEC61000-4-4 level 4 EFT stress (Voc = 2kV; Isc = 40A; 8/20µs).



Design Notes

1. This solution is targeted toward high-side current sensing.
2. The sense resistor value is determined by minimum and maximum load currents, power dissipation and Current Shunt Amplifier (CSA) gain.
3. Bidirectional current sensing requires an output reference voltage (Vref). Device gain is achieved through internal precision matched resitor network.
4. The expected maximum and minimum output voltage must be within the device linear range.
5. The TVS diode must be selected based on bus voltage, the CSA common-mode voltage specification, and EFT pulse characteristics.

Design Steps

1. Determine the maximum output swing:
   $VswN = Vref - VoMin = 2.5V - 0.1V = 2.4V$
   $VswP = VoMax - Vref = 4.9V - 2.5V = 2.4V$
2. Determine the maximum value of the sense resistor based on maximum load current, swing and device gain. In this example, a gain of 20 was chosen to illustrate the calculation, alternative gain versions may be selected as well:
   $Rshunt \le \frac{Vswp}{Iin\_max \times Gain} = \frac{2.4V}{40A \times 20} = 3m\,\Omega$
3. Calculate the peak power rating of the sense resistor:
   $Pshunt = Iin\_max^2 \times Rshunt = 40A^2 \times 3m\,\Omega = 5W$
4. Determine TVS standoff voltage and clamp voltage:
   $Vwm = 36V$ and $Vc \le 80V$
5. Select a TVS diode.

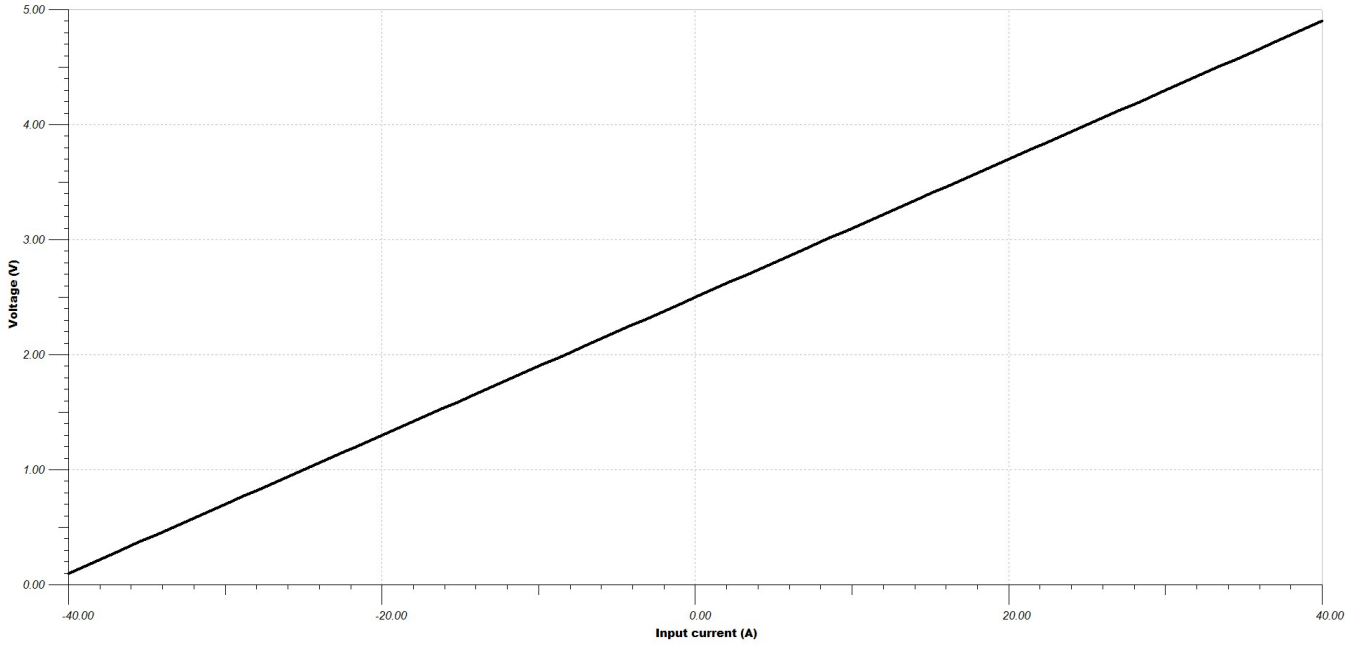
   For example, SMBJ36A from Littelfuse™ satisfies the previous requirement, with peak pulse power of 600W (10/1000µs) and current of 10.4A.
6. Make sure the TVS diode satisfies the design requirement based on the TVS operating curve.

   Peak pulse power at given excitation (8/20µs) is estimated to be around 3.5kW, which translates to peak pulse current:
   $Ipp = \frac{3.5kW}{600W} \times 10.4A = 60A$

This is above the maximum excitation (short circuit) current of 40A. The select TVS effectively protects the circuit against the specified EFT strike.

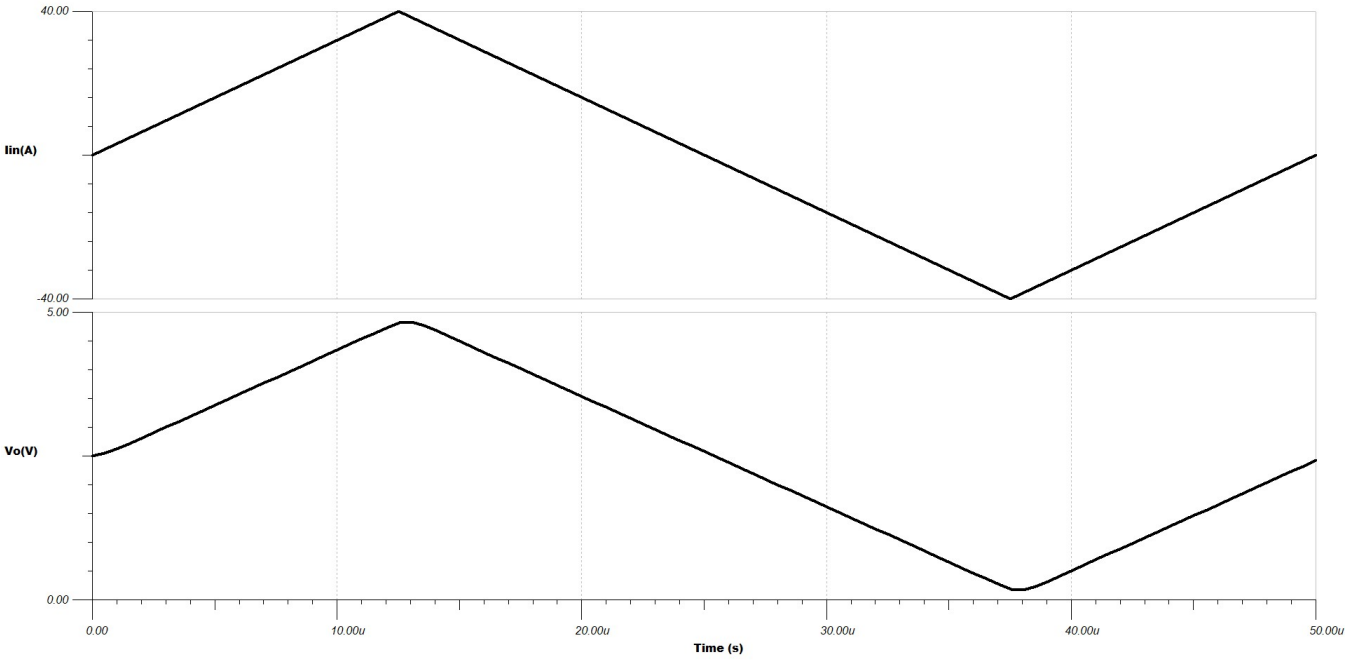
Design Simulations

DC Transfer Characteristics

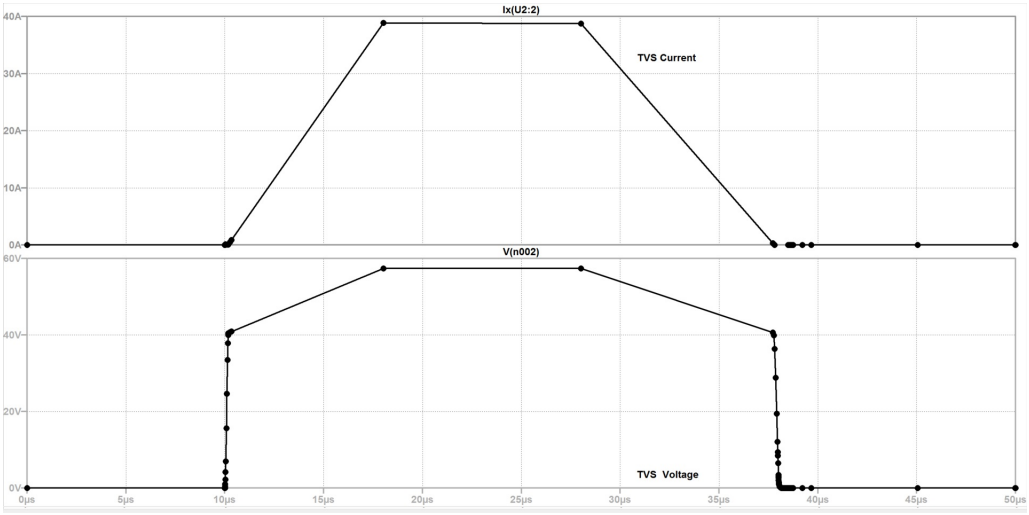


Transient Simulation Results

The output is a scaled version of the input.



TVS Diode Transient Response Under EFT Excitation



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

For more information on transient protection of the current sense amplifiers, see TIDA-00302 and the Current Sense Amplifier Training Videos.

Design Featured Current Sense Amplifier

INA240A1	
V_S	2.7V to 5.5V
V_{CM}	–4V to 80V
V_{os}	Rail-to-rail
V_{os}	5µV
I_B	80µA
BW	400kHz
Vos Drift	50nV/°C
http://www.ti.com/product/INA240	

Design Alternate

INA282	
V_S	2.7V to 18V
V_{CM}	–14V to 80V
V_{os}	20µV
I_B	25µA
BW	10kHz
Vos Drift	0.3µV/°C
http://www.ti.com/product/INA193	

Revision History

Revision	Date	Change
A	February 2019	Changed VinMin and VinMax in the Design Goals table to IinMin and IinMax, respectively.

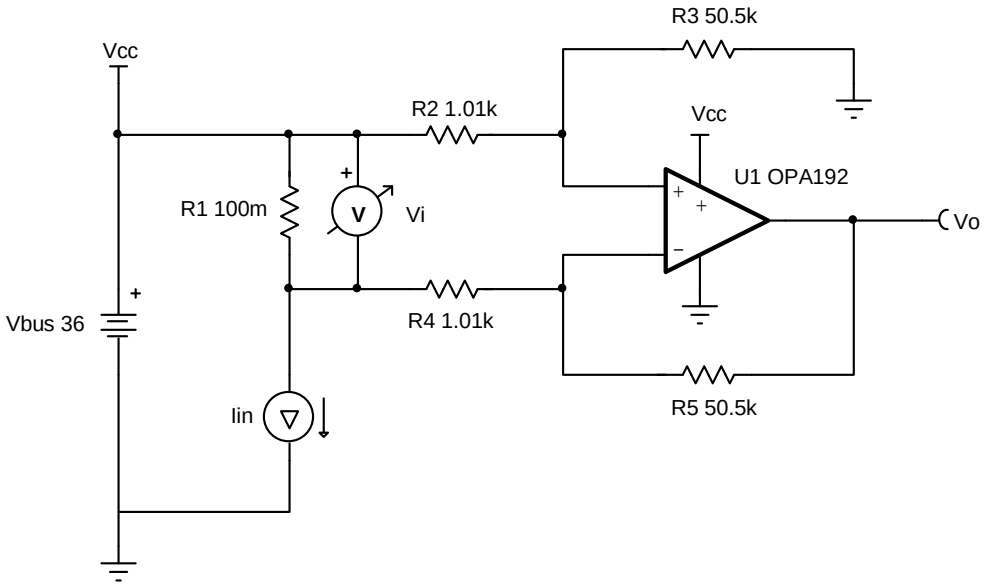
High-side current-sensing circuit design

Design Goals

Input		Output		Supply	
I_{iMin}	I_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}
50mA	1A	0.25V	5V	36V	0V

Design Description

This single–supply, high–side, low–cost current sensing solution detects load current between 50mA and 1A and converters it to an output voltage from 0.25V to 5V. High–side sensing allows for the system to identify ground shorts and does not create a ground disturbance on the load.



Design Notes

1. DC common mode rejection ratio (CMRR) performance is dependent on the matching of the gain setting resistors, R_2-R_5.
2. Increasing the shunt resistor increases power dissipation.
3. Ensure that the common–mode voltage is within the linear input operating region of the amplifier. The common mode voltage is set by the resistor divider formed by R_2, R_3, and the bus voltage. Depending on the common–mode voltage determined by the resistor divider a rail–to–rail input (RRI) amplifier may not be required for this application.
4. An op amp that does not have a common-mode voltage range that extends to V_{cc} may be used in low–gain or an attenuating configuration.
5. A capacitor placed in parallel with the feedback resistor will limit bandwidth, improve stability, and help reduce noise.
6. Use the op amp in a linear output operating region. Linear output swing is usually specified under the A_{OL} test conditions.

Design Steps

1. The full transfer function of the circuit is provided below.

$$V_o = I_{in} \times R_1 \times \frac{R_5}{R_4}$$

Given $R_2 = R_4$ and $R_3 = R_5$

2. Calculate the maximum shunt resistance. Set the maximum voltage across the shunt to 100mV.

$$R_1 = \frac{V_{iMax}}{I_{iMax}} = \frac{100mV}{1A} = 100m\Omega$$

3. Calculate the gain to set the maximum output swing range.

$$Gain = \frac{V_{oMax} - V_{oMin}}{(I_{iMax} - I_{iMin}) \times R_1} = \frac{5V - 0.25V}{(1A - 0.05A) \times 100m\Omega} = 50\frac{V}{V}$$

4. Calculate the gain setting resistors to set the gain calculated in step 3.

Choose $R_2 = R_4 = 1.01k\,\Omega$ (Standard value)

$$R_3 = R_5 = R_2 \times Gain = 1.01k\,\Omega \times 50\frac{V}{V} = 50.5k\,\Omega \text{ (Standard value)}$$

5. Calculate the common–mode voltage of the amplifier to ensure linear operation.

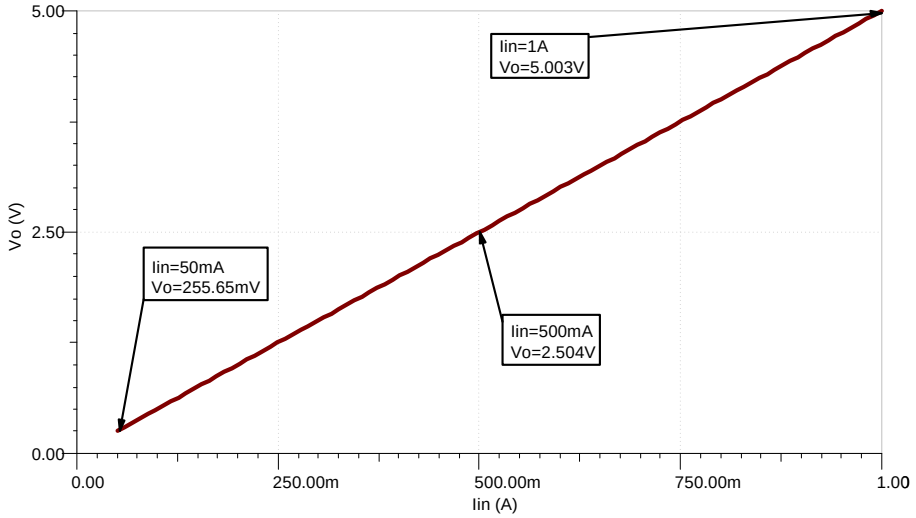
$$V_{cm} = V_{CC} \times \frac{R_3}{R_2 + R_3} = 36V \times \frac{50.5k}{1.01k + 50.5k} = 35.294\,V$$

6. The upper cutoff frequency (f_H) is set by the non–inverting gain (noise gain) of the circuit and the gain bandwidth (GBW) of the op amp.

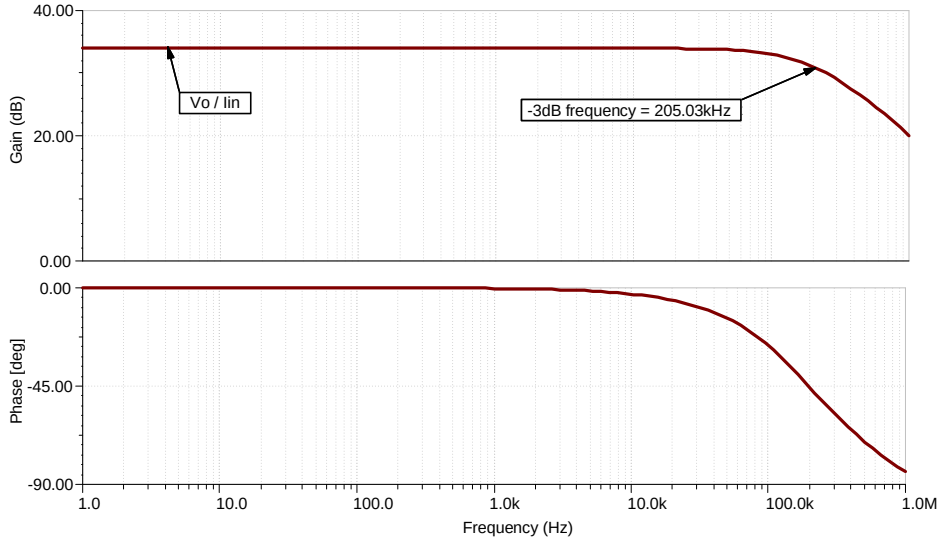
$$f_H = \frac{GBW}{Noise\ Gain} = \frac{10MHz}{51\frac{V}{V}} = 196.1\ kHz$$

Design Simulations

DC Simulation Results



AC Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAV4
3. TI Precision Labs

Design Featured Op Amp

OPA192	
V_{cc}	4.5V to 36V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	5µV
I_q	1mA
I_b	5pA
UGBW	10MHz
SR	20V/µs
#Channels	1, 2, 4
www.ti.com/product/OPA192	

Design Alternate Op Amp

OPA2990	
V_{cc}	2.7V to 40V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	250µV
I_q	120µA
I_b	10pA
UGBW	1.25MHz
SR	5V/µs
#Channels	2
www.ti.com/product/OPA2990	

Revision History

Revision	Date	Change
A	February 2019	Downstyle title. Added Design Alternate Op Amp table.

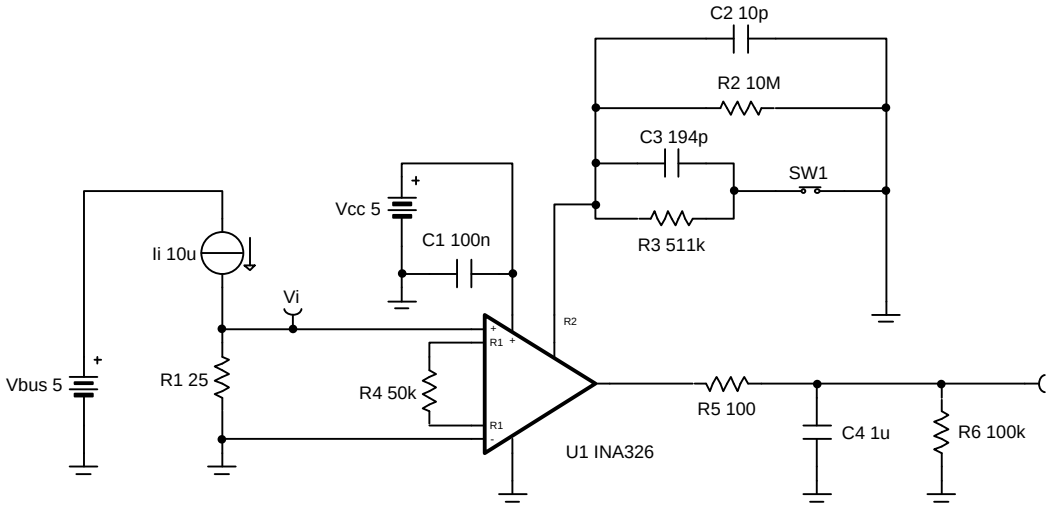
3-decade, load-current sensing circuit

Design Goals

Input		Output		Supply		
I_{iMin}	I_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
10µA	10mA	100mV	4.9V	5.0V	0V	0V

Design Description

This single-supply, low-side, current-sensing solution accurately detects load current between 10µA and 10mA. A unique yet simple gain switching network was implemented to accurately measure the three-decade load current range.



Design Notes

1. Use a maximum shunt resistance to minimize relative error at minimum load current.
2. Select 0.1% tolerance resistors for R_1, R_2, R_3, and R_4 in order to achieve approximately 0.1% FSR gain error.
3. Use a switch with low on-resistance (R_{on}) to minimize interaction with feedback resistances, preserving gain accuracy.
4. Minimize capacitance on INA326 gain setting pins.
5. Scale the linear output swing based on the gain error specification.

Design Steps

1. Define full-scale shunt resistance.

$$R_1 = \frac{V_{iMax}}{I_{iMax}} = \frac{250mV}{10mA} = 25\Omega$$

2. Select gain resistors to set output range.

$$G_{IiMax} = \frac{V_{oMax}}{V_{iMax}} = \frac{V_{oMax}}{R_1 \times I_{iMax}} = \frac{4.9V}{25\Omega \times 10mA} = 19.6\frac{V}{V}$$

$$G_{IiMin} = \frac{V_{oMin}}{V_{iMin}} = \frac{V_{oMin}}{R_1 \times I_{iMin}} = \frac{100mV}{25\Omega \times 10\mu A} = 400\frac{V}{V}$$

$$R_2 = \frac{R_4 \times G_{IiMin}}{2} = \frac{50k\Omega \times 400\frac{V}{V}}{2} = 10M\Omega$$

$$R_2 \parallel R_3 = \frac{R_4 \times G_{IiMax}}{2} = \frac{50k\Omega \times 19.6\frac{V}{V}}{2} = 490k\Omega$$

$$R_3 = \frac{490k\Omega \times R_2}{R_2 - 490k\Omega} = 515.25k\Omega \approx 511k\Omega \text{ (Standard Value)}$$

3. Select a capacitor for the output filter.

$$f_p = \frac{1}{2 \times \pi \times R_5 \times C_4} = \frac{1}{2 \times \pi \times 100\Omega \times 1\ \mu F} = 1.59kHz$$

4. Select a capacitor for gain and filtering network.

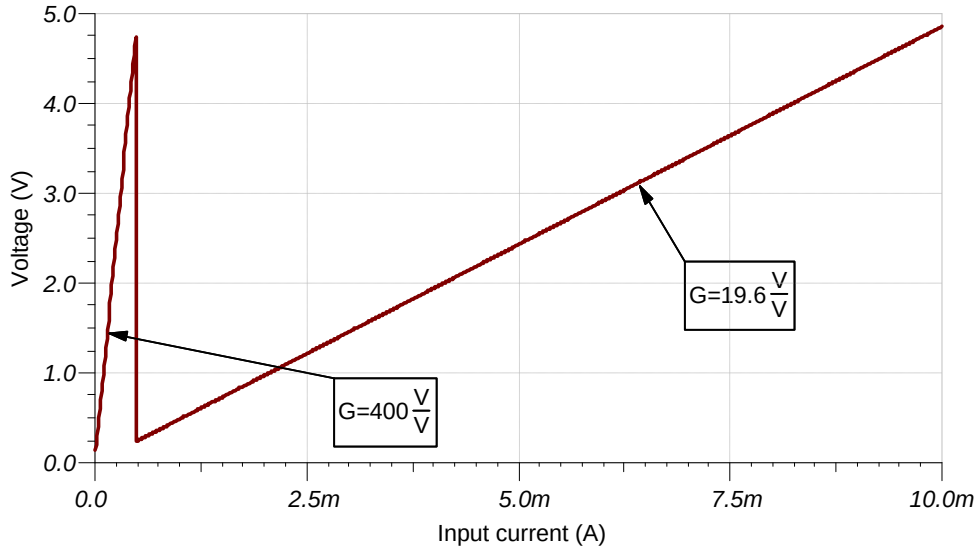
$$C_2 = \frac{1}{2 \times \pi \times R_2 \times f_p} = \frac{1}{2 \times \pi \times 10M\Omega \times 1.59kHz} = 10pF$$

$$C_3 = \frac{1}{2 \times \pi \times (R_2 || R_3) \times f_p} - C_2 = \frac{1}{2 \times \pi \times (10M\Omega || 511k\Omega) \times 1.59kHz} - 10pF$$

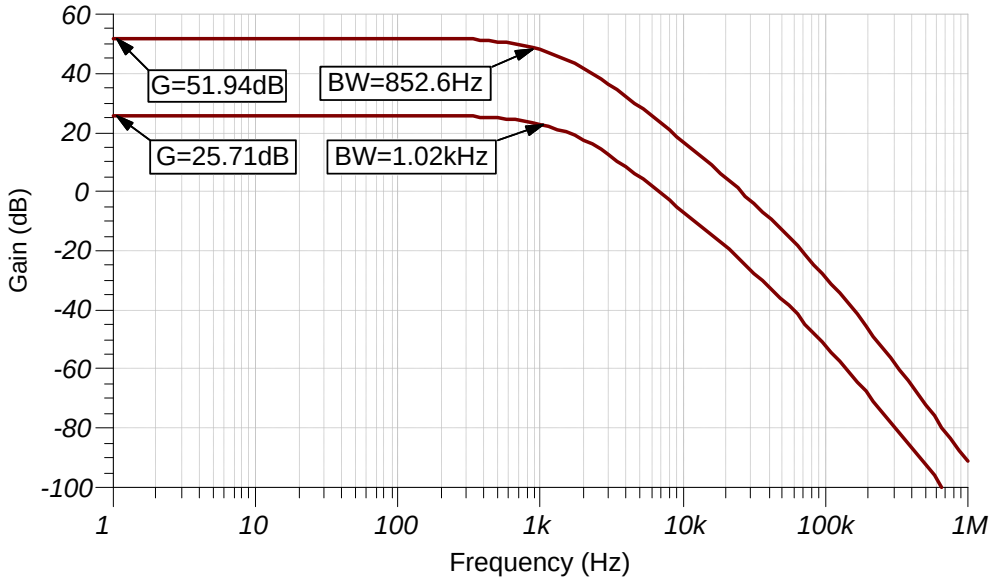
$$C_3 = 196pF \approx 194pF \text{ (Standard Value)}$$

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC498.

See TIPD104, www.ti.com/tool/tipd104.

Design Featured Op Amp

INA326	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.1mV
I_q	3.4mA
I_b	2nA
UGBW	1kHz
SR	Filter limited
#Channels	1
www.ti.com/product/ina326	

Revision History

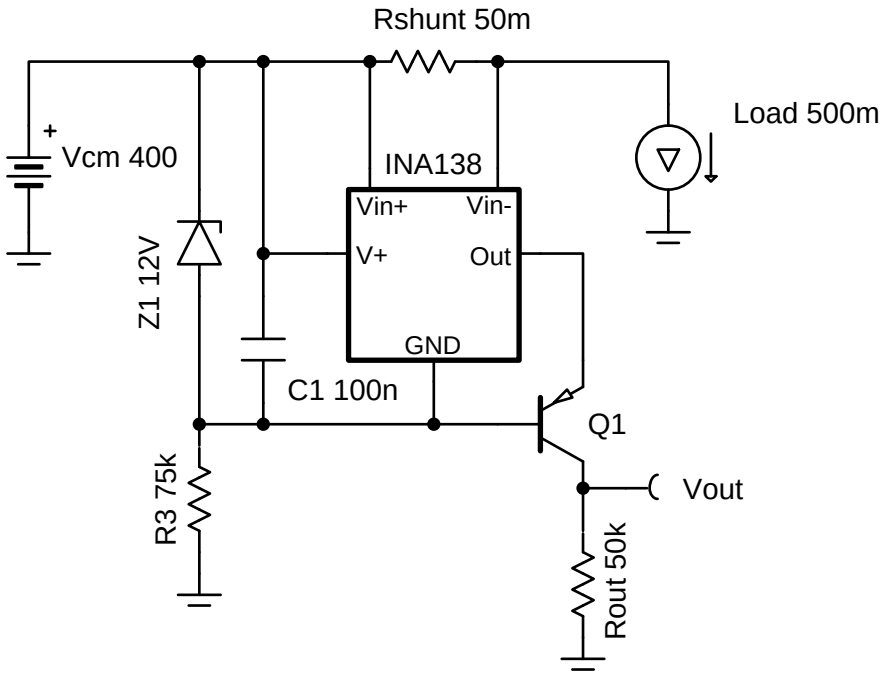
Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

High-voltage, high-side floating current sensing circuit using current output, current sense amplifier

Input		Output		Supply		
$I_{load\ Min}$	$I_{load\ Max}$	$V_{out\ Min}$	$V_{out\ Max}$	$V_{cm\ Min}$	$V_{cm\ Max}$	V_{ee}
0.5A	9.9A	250mV	4.95V	12V	400V	GND (0V)

Design Description

This cookbook is intended to demonstrate a method of designing an accurate current sensing solution for systems with high common mode voltages. The principle aspect of this design uses a unidirectional circuit to monitor a system with V_{cm} = 400V by floating the supplies of the device across a Zener diode from the supply bus (V_{cm}). This cookbook is based on the High Voltage 12 V – 400 V DC Current Sense Reference Design.



Design Notes

1. The Getting Started with Current Sense Amplifiers video series introduces implementation, error sources, and advanced topics for using current sense amplifiers.
2. This example is for high V_{CM}, high-side, unidirectional, DC sensing.
3. To minimize error, make the shunt voltage as large as the design will allow. For the INA138 device, keep V_{sense} >> 15mV.
4. The relative error due to input offset increases as shunt voltage decreases, so use a current sense amplifier with low offset voltage. A precision resistor for R_{shunt} is necessary because R_{shunt} is a major source of error.
5. The INA138 is a current-output device, so voltages referenced to ground are achieved with a high voltage bipolar junction transistor (BJT).
   - Ensure the transistor chosen for Q1 can withstand the maximum voltage across the collector and emitter (for example, need 400V, but select > 450V for margin).
   - Multiple BJTs can be stacked and biased in series to achieve higher voltages
   - High beta of this transistor reduces gain error from current that leaks out of the base

Design Steps

1. Determine the operating load current and calculate R_{shunt}:
   - Recommended V_{sense} is 100mV and maximum recommended is 500mV, so the following equation can be used to calculate R_{shunt} where $V_{sense} \leq 500mV$:

$$R_{shunt} = \frac{V_{sense\ max}}{I_{load\ max}} \rightarrow \frac{0.5V}{10A} = 50m\Omega$$

   - For more accurate and precise measurements over the operating temperature range, a current monitor with integrated shunt resistor can be used in some systems. The benefits of using these devices are explained in Getting Started with Current Sense Amplifiers, Session 16: Benefits of Integrated Precision Shunt Resistor.
2. Choose a Zener diode to create an appropriate voltage drop for the INA138 supply:
   - The Zener voltage of the diode should fall in the INA138 supply voltage range of 2.7V to 36V and needs to be larger than the maximum output voltage required.
   - The Zener diode voltage regulates the INA138 supply and protects from transients.
   - Data sheet parameters are defined for 12-V V_{in+} to the GND pin so a 12-V Zener is chosen.
3. Determine the series resistance with the Zener diode:
   - This resistor (R3) is the main power consumer due to its voltage drop (up to 388V in this case). If R3 is too low, it will dissipate more power, but if it is too high R3 will not allow the Zener diode to avalanche properly. Since the data sheet specifies I_Q for V_S = 5V, estimate the max quiescent current of the INA138 device at V_S = 12V to be 108µA and calculate R3 using the bias current of the Zener diode, 5mA, as shown:

$$R_3 = \frac{V_{CM} - V_{zener}}{I_{zener} + I_{INA138}} = \frac{400V - 12V}{5mA + 108\mu A} \approx 75.96k\Omega$$

$$\text{standard value} \rightarrow 75k\Omega$$

   - The power consumption of this resistor is calculated using the following equation:

$$Power_{R3} = \frac{(V_{cm} - V_{Zener})^2}{R3} \rightarrow \frac{(400V - 12V)^2}{75k\Omega} \approx 2.007W$$

4. Calculate R_{out} using the equation for output current in the INA138 data sheet.
   - This system is designed for 10V/V gain where V_{out} = 1V if V_{sense} = 100mV:

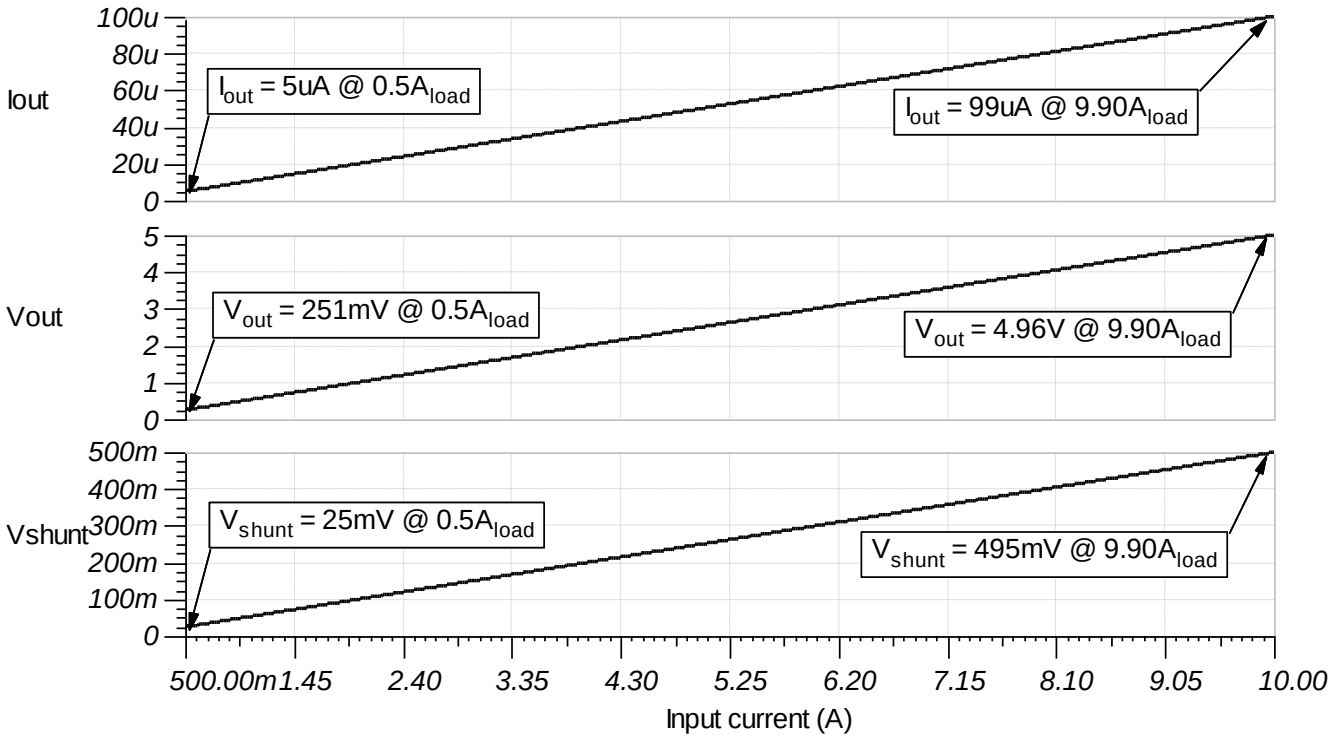
$$I_{out\ INA138} = 200\frac{\mu A}{V} \times (V_{sense\ max}) \rightarrow 200\frac{\mu A}{V} \times (0.5V) = 100\mu A$$

$$R_{out} = \frac{V_{out\ max}}{I_{out\ INA138}} \rightarrow \frac{5V}{100\mu A} = 50k\Omega$$

Design Simulations

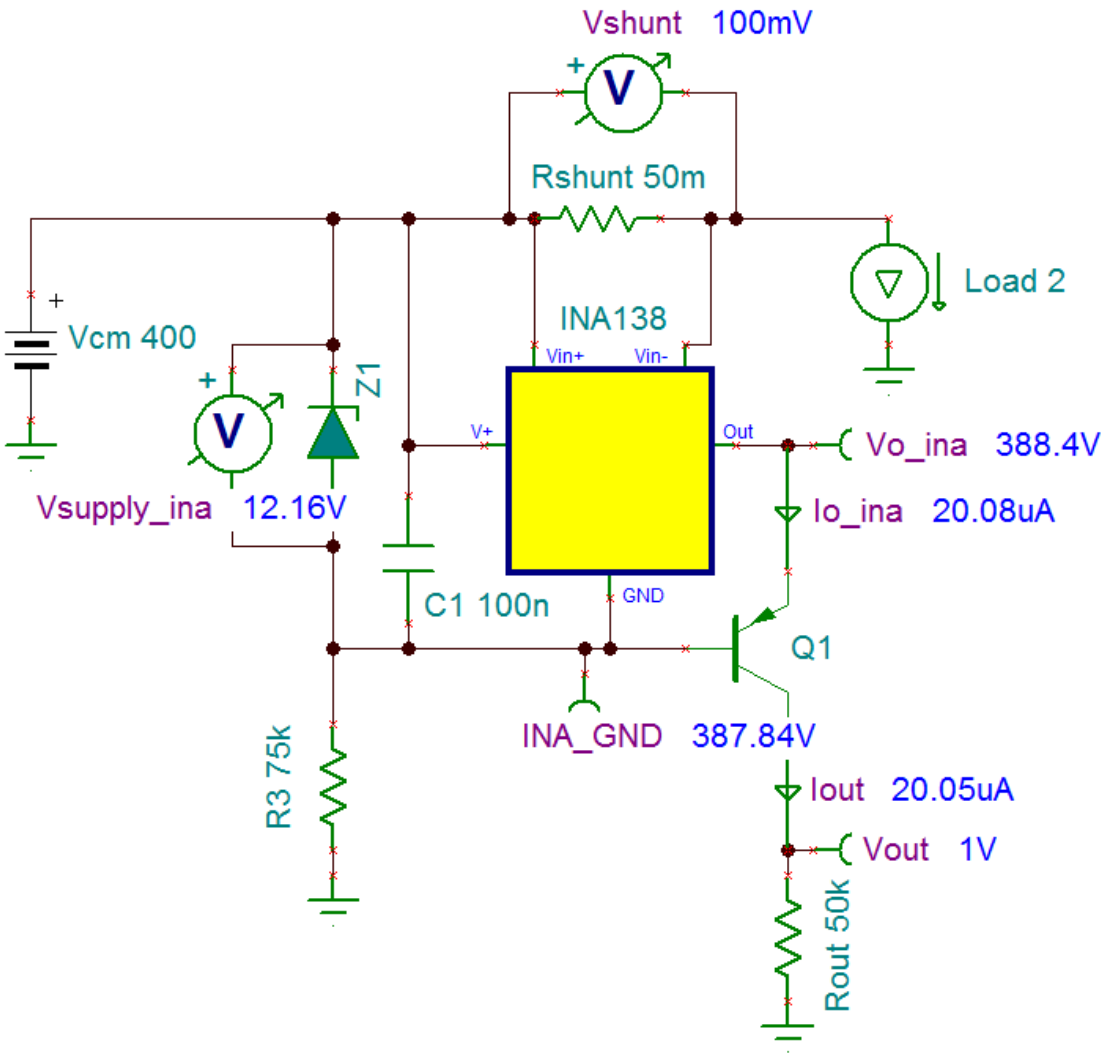
DC Simulation Results

The following graph shows a linear output response for load currents from 0.5A to 10A and 12V ≤ V_{cm} ≤ 400V. I_{out} and V_{out} remain constant over a varying V_{cm} once the Zener diode is reverse biased.



Steady State Simulation Results

The following image shows this system in DC steady state with a 2-A load current. The output voltage is 10× greater than the measured voltage across R_{shunt}.



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SGLC001.

Getting Started with Current Sense Amplifiers video series:

https://training.ti.com/getting-started-current-sense-amplifiers

Abstract on Extending Voltage Range of Current Shunt Monitor:

http://www.ti.com/lit/an/slla190/slla190.pdf

High Voltage 12V – 400V DC Current Sense Reference Design:

http://www.ti.com/tool/TIDA-00332

Cookbook Design Files:

http://proddms.itg.ti.com/stage/lit/sw/sglc001a/sglc001a.zip

Current Sense Amplifiers on TI.com:

http://www.ti.com/amplifier-circuit/current-sense/products.html

For direct support from TI Engineers use the E2E community:

http://e2e.ti.com

Design Featured Current Shunt Monitor

INA138	
V_{ss}	2.7V to 36V
$V_{in\ cm}$	2.7V to 36V
V_{out}	Up to (V+) - 0.8V
V_{os}	±0.2mV to ±1mV
I_q	25µA to 45 µA
I_b	2 µA
UGBW	800kHz
# of Channels	1
http://www.ti.com/product/ina138	

Design Alternate Current Shunt Monitor

INA168	
V_{ss}	2.7V to 60V
$V_{in\ cm}$	2.7V to 60V
V_{out}	Up to (V+) - 0.8V
V_{os}	±0.2mV to ±1mV
I_q	25µA to 45 µA
I_b	2 µA
UGBW	800kHz
# of Channels	1
http://www.ti.com/product/ina168	

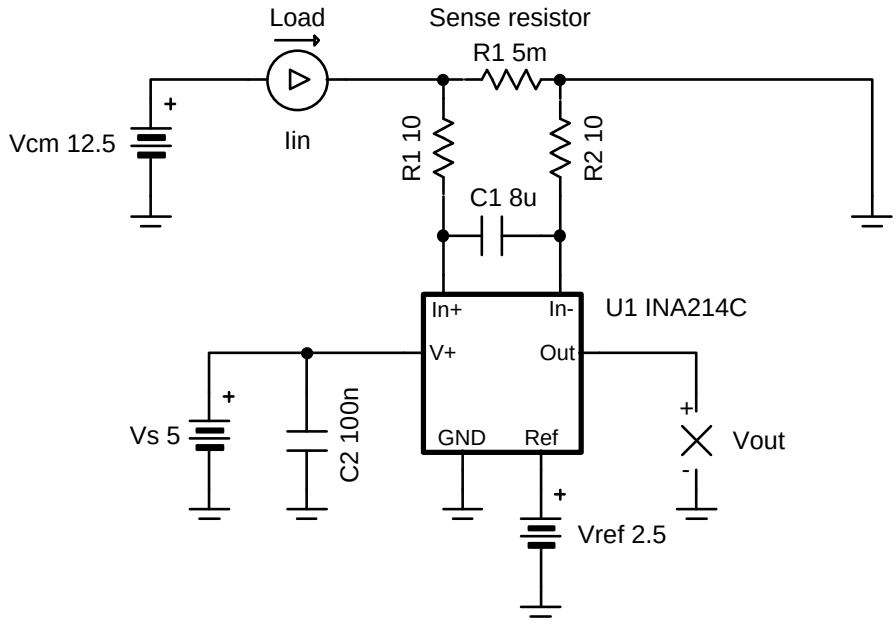
Low-drift, low-side, bidirectional current sensing circuit with integrated precision gain resistors

Design Goals

Input			Output		Supply	
I_{inMin}	I_{inMax}	V_{cm}	V_{outMin}	V_{outMax}	V_s	V_{ref}
–4A	4A	12.5V	0.5V	4.5V	5V	2.5V

Design Description

The low-side bidirectional current-shunt monitor solution illustrated in the following image can accurately measure currents from –4A to 4A, and the design parameters can easily be changed for different current measurement ranges. Current-shunt monitors from the INA21x family have integrated precision gain resistors and a zero-drift architecture that enables current sensing with maximum drops across the shunt as low as 10mV full-scale.



Design Notes

- To avoid additional error, use $R_1 = R_2$ and keep the resistance as small as possible (no more than 10Ω, as stated in INA21x Voltage Output, Low- or High-Side Measurement, Bidirectional, Zero-Drift Series, Current-Shunt Monitors).
- Low-side sensing should not be used in applications where the system load cannot withstand small ground disturbances or in applications that need to detect load shorts.
- The Getting Started with Current Sense Amplifiers video series introduces implementation, error sources, and advanced topics that are good to know when using current sense amplifiers.

Design Steps

1. Determine V_{ref} based on the desired current range:

   With a current range of –4A to 4A, then half of the range is below 0V, so set:

   $$V_{ref} = \frac{1}{2} V_s = \frac{5}{2} = 2.5V$$

2. Determine the desired shunt resistance based on the maximum current and maximum output voltage:

   To not exceed the swing-to-rail and to allow for some margin, use V_{outMax} = 4.5V. This, combined with maximum current of 4A and the V_{ref} calculated in step 1, can be used to determine the shunt resistance using the equation:

   $$R_1 = \frac{V_{outMax} - V_{ref}}{Gain \times I_{loadMax}} = \frac{4.5 - 2.5}{100 \times 4} = 5m\Omega$$

3. Confirm V_{out} will be within the desired range:

   At the maximum current of 4A, with Gain = 100V/V, R_1 = 5mΩ, and V_{ref} = 2.5V:

   $$V_{out} = I_{load} \times Gain \times R_1 + V_{ref} = 4 \times 100 \times 0.005 + 2.5 = 4.5V$$

   At the minimum current of 4A, with Gain = 100V/V, R_1 = 5mΩ, and V_{ref} = 2.5V:

   $$V_{out} = I_{load} \times Gain \times R_1 + V_{ref} = -4 \times 100 \times 0.005 + 2.5 = 0.5V$$

4. Filter cap selection:

   To filter the input signal at 1kHz, using $R_1 = R_2$ = 10Ω:

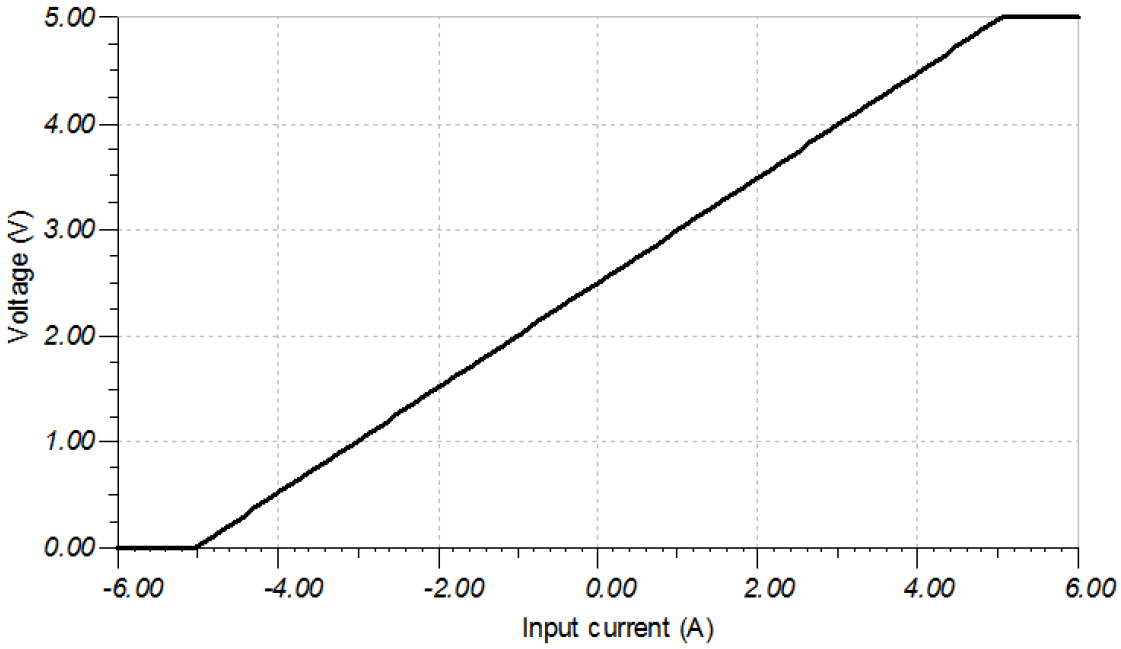
   $$C_1 = \frac{1}{2\pi(R_1 + R_2)F_{-3dB}} = \frac{1}{2\pi(10+10)1000} = 7.958 \times 10^{-6} \approx 8\mu F$$

For more information on signal filtering and the associated gain error, see INA21x Voltage Output, Low- or High-Side Measurement, Bidirectional, Zero-Drift Series, Current-Shunt Monitors.

Design Simulations

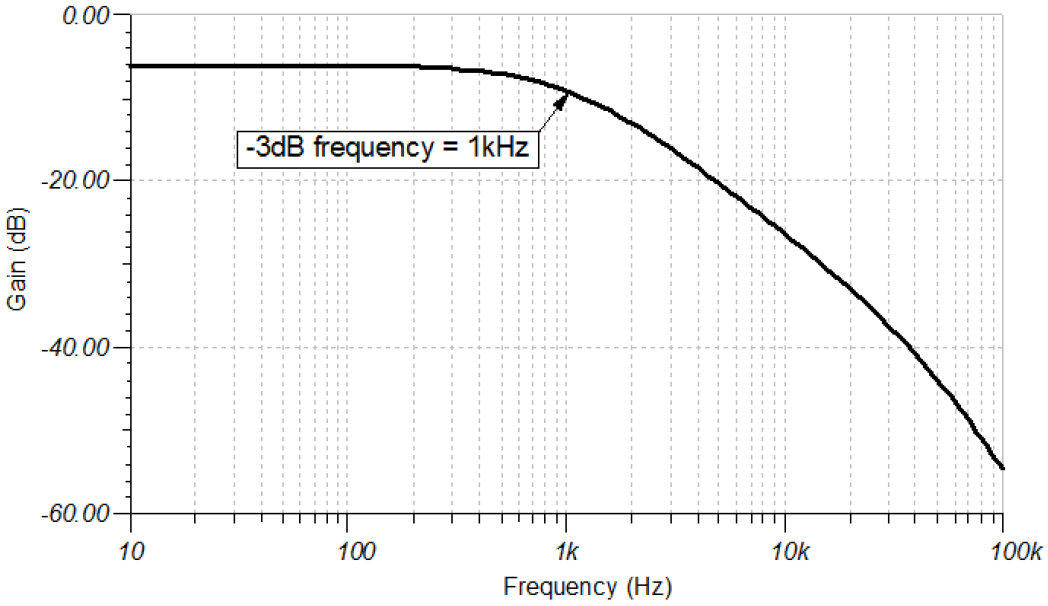
DC Analysis Simulation Results

The following plot shows the simulated output voltage V_{out} for the given input current I_{in}.



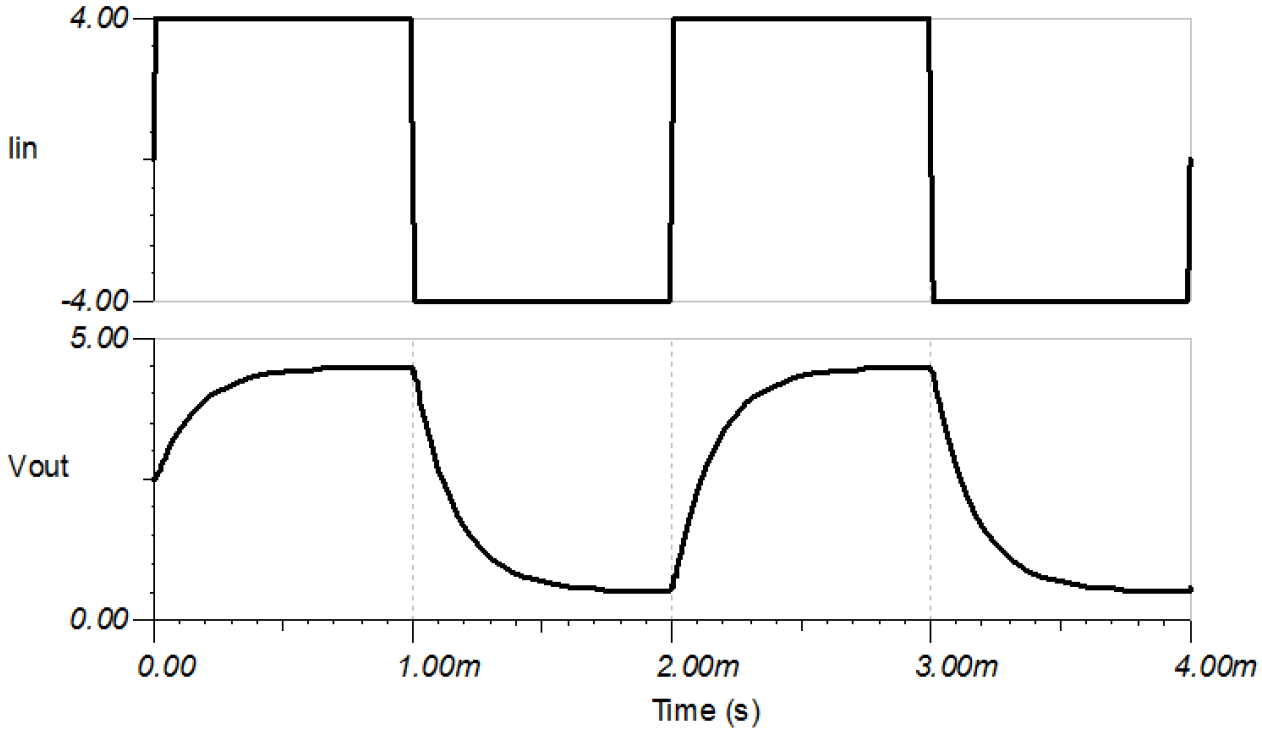
AC Analysis Simulation Results

The following plot shows the simulated gain vs frequency, as designed for in the design steps.



Transient Analysis Simulation Results

The following plot shows the simulated delay and settling time of the output V_{out} for a step response in I_{in} from –4A to 4A.



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

Circuit SPICE simulation File: http://proddms.itg.ti.com/fnview/sboc518

Getting Started with Current Sense Amplifiers video series: https://training.ti.com/getting-started-current-sense-amplifiers

Current Sense Amplifiers on TI.com: http://www.ti.com/amplifier-circuit/current-sense/products.html

For direct support from TI Engineers use the E2E community: http://e2e.ti.com

Design Featured Current Sense Amplifier

INA214C	
V_s	2.7V to 26V
V_{cm}	GND-0.1V to 26V
V_{out}	GND-0.3V to V_s+0.3V
V_{os}	±1µV typical
I_q	65µA typical
I_b	28µA typical
http://www.ti.com/product/INA214	

Design Alternate Current Sense Amplifiers

INA199C	
V_s	2.7V to 26V
V_{cm}	GND-0.1V to 26V
V_{out}	GND-0.3V to V_s+0.3V
V_{os}	±5µV typical
I_q	65µA typical
I_b	28µA typical
http://www.ti.com/product/INA199	

INA181	
V_s	2.7V to 5.5V
V_{cm}	GND-0.2V to 26V
V_{out}	GND-0.3V to V_s+0.3V
V_{os}	±100µV typical
I_q	65µA typical
I_b	195µA typical
http://www.ti.com/product/INA181	

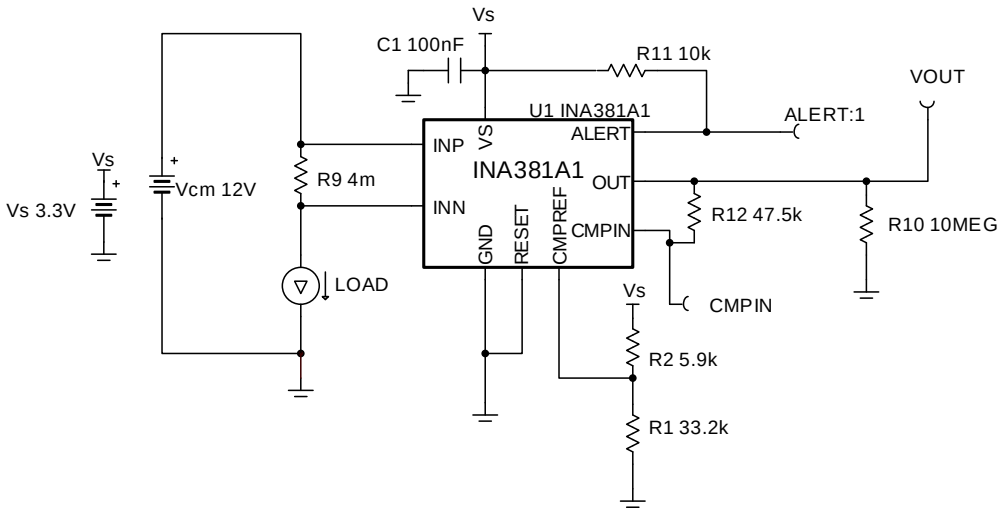
Overcurrent event detection circuit

Design Goals

Input		Overcurrent Conditions		Output		Supply	
$I_{load\ Min}$	$I_{load\ Max}$	I_{OC_TH}	$I_{Release\_TH}$	V_{out_OC}	$V_{out\_release}$	V_S	V_{REF}
1.5A	40A	35A	32A	2.8V	2.61V	3.3V	2.8V

Design Description

This is a unidirectional current sensing solution generally referred to as overcurrent protection (OCP) that can provide an overcurrent alert signal to shut off a system for a threshold current and re-engage the system once the output drops below a desired voltage ($V_{out\_release}$) lower than the overcurrent output threshold voltage (V_{out_OC}). In this particular setup, the sensing range is from 1.5A to 40A, with the overcurrent threshold defined at 35A (I_{OC_TH}). The system re-asserts the ALERT to high once the current has dropped below 32A ($I_{Release\_TH}$). The current shunt monitor is powered from a 3.3-V supply rail. OCP can be applied to both high-side and low-side topologies. The solution presented in this article is a high-side implementation.



Design Notes

1. Use low-tolerance, high-precision resistors if using a voltage divider for CMPREF and consider buffering the voltage. Otherwise consider using a low-dropout regulator (LDO), reference or buffered reference voltage circuit to supply the CMPREF.
2. Use decoupling capacitors to ensure the device supply is stable, such as C1. Also place the decoupling capacitor as close to the device supply pin as possible.

Design Steps

1. Calculate the R_{shunt} value given 20V/V gain. Use the nearest standard value shunt, preferably lower than the calculated shunt to avoid railing the output prematurely .

$$R_{shunt} = \frac{V_{out\ max}}{gain \times I_{max}} = \frac{V_S - 0.02V}{gain \times I_{max}} = \frac{3.3V - 0.02V}{20V/V \times 40A} = 0.0041\Omega$$

$$R_{standard\ \ shunt} = 4m\Omega\ (standard\ 1\%\ value)$$

2. Determine the voltage at the current shunt monitor output for the overcurrent threshold.

$$V_{out\_35A} = I_{OC\_TH} \times R_{standard\ \ shunt} \times gain = 35A \times 4m\Omega \times 20V/V = 2.8V$$

3. Choose a standard resistor value for R_1 and solve for R_2.

   A resistor with kilo-ohm resistance or higher is desired to minimize power loss. Through calculation, 33.2kΩ and 5.9kΩ were chosen for resistances $R_1$and R_2.

$$R_2 = \left(\frac{V_S}{V_{out\_35A}} - 1\right) \times R_1 = \left(\frac{3.3V}{2.8V} - 1\right) \times 33.2k\Omega = 5.9k\Omega$$

4. Calculate the resistance (R_{Hyst}) required for the proper hysteresis.

$$R_{Hyst} = \frac{V_{out\_35A} - (I_{Release\ \_TH} \times R_{standard\ \ shunt} \times gain + V_{Hyst\_def})}{I_{Hyst}}$$

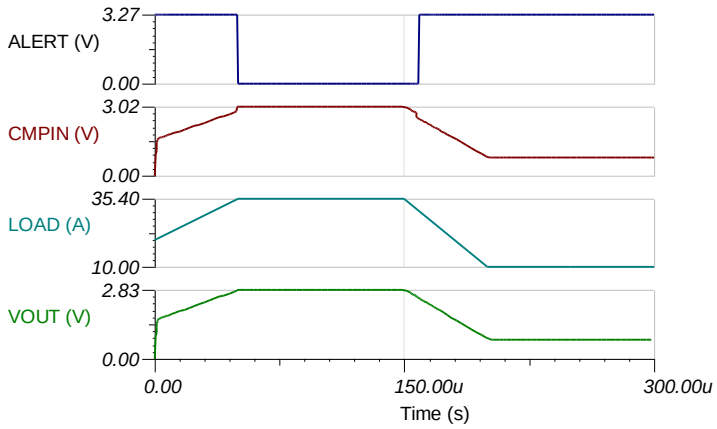
$$R_{Hyst} = \frac{2.8V - (32A \times 4m\Omega \times 20V/V + 50mV)}{4\mu A} = 47.5k\Omega$$

Design Simulations

Transient Simulation Results

Considering error, V_{out_OC} is expected to be approximately 2.8V, while $V_{out\_release}$ is expected to be approximately 2.61V.

High-Side OCP Simulation Results



The device exhibits an active low on the Alert pin when the load reaches 35A and re-asserts Alert to high when the load drops below 32A. If the user zooms in and looks at the VOUT voltage, and accounts for an expected propagation delay of 0.4µs, the device output is 2.69V at I_{OC_TH}, which only has an error of 0.39% with respect to the ideal output of 2.8V. At $I_{release\_TH}$, the alert re-asserts to high when the output dropped to 2.58V, which only has an error of 1.15% with respect to the ideal output of 2.61V.

Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

Key files for Overcurrent Protection Circuit:

Source files for this design:
High-Side OCP Tina Model
Low-Side OCP Tina Model

Getting Started with Current Sense Amplifiers video series:

https://training.ti.com/getting-started-current-sense-amplifiers

Design Featured Current Sense Amplifier

INA381	
V_S	2.7V to 5.5V
V_{CM}	GND-0.3V to 26V
V_{OUT}	GND+5μV to V_S-0.02V
V_{OS}	±100 typical
I_q	250μA typical
I_B	80μA typical
http://www.ti.com/product/INA381	

Design Alternate Current Sense Monitor

	INA301	INA302	INA303
V_S	2.7V to 5.5V	2.7V to 5.5V	2.7V to 5.5V
V_{CM}	GND-0.3V to 40V	-0.1V to 36V	-0.1V to 36V
V_{OUT}	GND+0.02 to V_S-0.05V	GND+0.015 to V_S-0.05V	GND+0.015 to V_S-0.05V
V_{OS}	Gain Dependent	Gain Dependent	Gain Dependent
I_q	500μA typical	850μA typical	850μA typical
I_B	120μA typical	115μA typical	115μA typical
Comparator	Single Comparators	Dual Comparators	Window Comparators
	http://www.ti.com/product/INA301	http://www.ti.com/product/INA302	http://www.ti.com/product/INA303

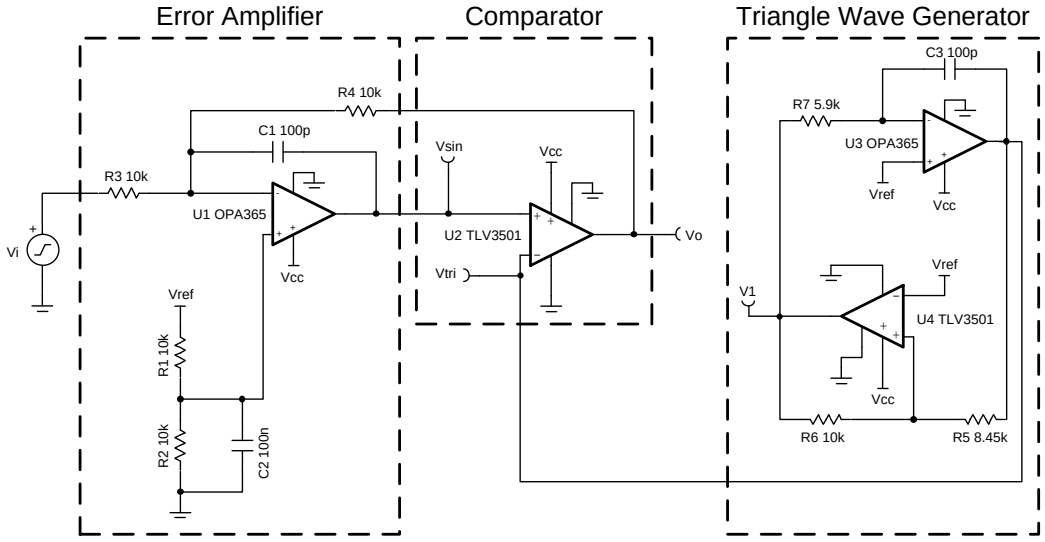
PWM generator circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–2.0V	2.0V	0V	5V	5V	0V	2.5V

Design Description

This circuit utilizes a triangle wave generator and comparator to generate a 500 kHz pulse-width-modulated (PWM) waveform with a duty cycle that is inversely proportional to the input voltage. An op amp and comparator (U_3 and U_4) generate a triangle waveform which is applied to the inverting input of a second comparator (U_2). The input voltage is applied to the non-inverting input of U_2. By comparing the input waveform to the triangle wave, a PWM waveform is produced. U_2 is placed in the feedback loop of an error amplifier (U_1) to improve the accuracy and linearity of the output waveform.



Design Notes

1. Use a comparator with push-pull output and minimal propagation delay.
2. Use an op amp with sufficient slew rate, GBW, and voltage output swing.
3. Place the pole created by C_1 below the switching frequency and well above the audio range.
4. V_{ref} must be low impedance (for example, output of an op amp).

Design Steps

1. Set the error amplifier inverting signal gain.

   $\text{Gain} = -\frac{R_4}{R_3} = -1\frac{V}{V}$

   Select $R_3 = R_4 = 10k\Omega$

2. Determine R_1 and R_2 to divide V_{ref} to cancel the non-inverting gain.

   $V_{o\_dc} = (1 + \frac{R_4}{R_3})(\frac{R_2}{R_1 + R_2}) \times Vref$

   $R_1 = R_2 = R_3 = R_4 = 10k\Omega$, $V_{o\_dc} = 2.5V$

3. The amplitude of V_{tri} must be chosen such that it is greater than the maximum amplitude of V_i (2.0V) to avoid 0% or 100% duty cycle in the PWM output signal. Select V_{tri} to be 2.1V. The amplitude of V_1 = 2.5V.

   $V_{tri}\,(\text{Amplitude}) = \frac{R_5}{R_6} \times V_1(\text{Amplitude})$

   Select R_6 to be $10k\Omega$, then compute R_5

   $R_5 = \frac{V_{tri}(\text{Amplitude}) \times R_6}{V_1\ (\text{Amplitude})} = 8.4k\Omega \approx 8.45k\Omega$ (Standard Value)

4. Set the oscillation frequency to 500kHz.

   $f_t = \frac{R_6}{4 \times R_7 \times R_5 \times C_3}$

   Set $C_3 = 100pF$, then compute R_7

   $R_7 = \frac{R_6}{4 \times f_t \times R_5 \times C_3} = 5.92k\Omega \approx 5.90k\Omega$ (Standard Value)

5. Choose C_1 to limit amplifier bandwidth to below switching frequency.

   $f_p = \frac{1}{2 \times \pi \times R_4 \times C_1}$

   $C_1 = 100pF \rightarrow f_p = 159kHz$

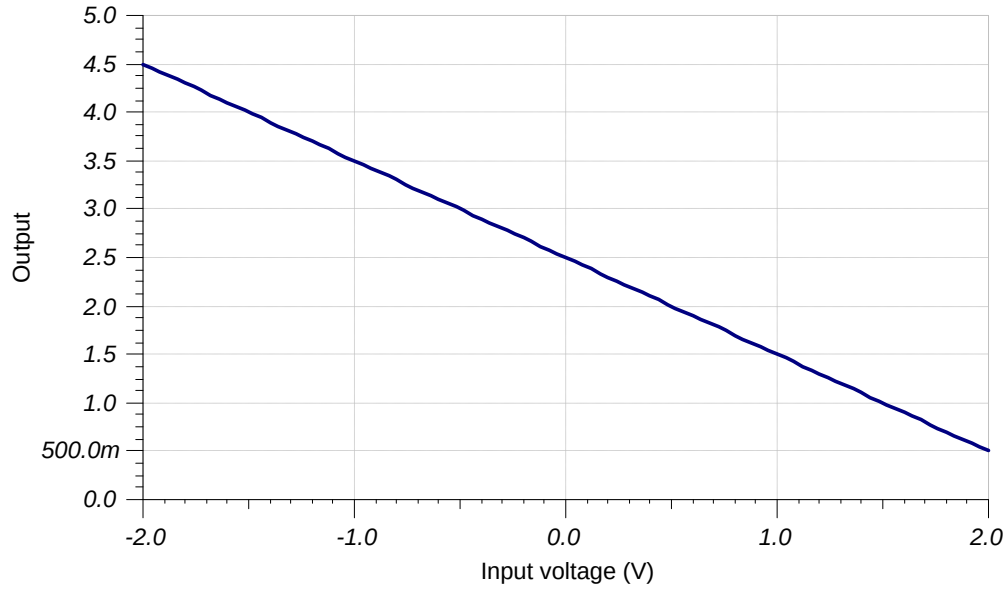
6. Select C_2 to filter noise from V_{ref}.

   $C_2 = 100nF$ (Standard Value)

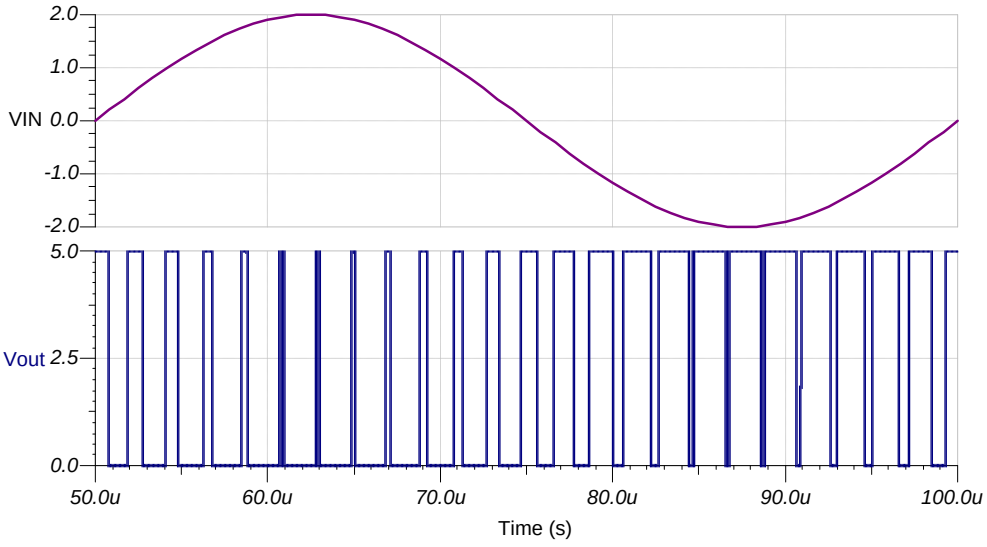
   $f_{div} = \frac{1}{2 \times \pi \times C_2 \times \frac{R_1 \times R_2}{R_1 + R_2}} = 320Hz$

Design Simulations

DC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC502.

See TIPD108, www.ti.com/tool/tipd108

Design Featured Op Amp

OPA2365	
V_{ss}	2.2V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	100µV
I_q	4.6mA
I_b	2pA
UGBW	50MHz
SR	25V/µs
#Channels	2
www.ti.com/product/opa2365	

Design Comparator

TLV3502	
V_{ss}	2.2V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1mV
I_q	3.2mA
I_b	2pA
UGBW	-
SR	-
#Channels	2
www.ti.com/product/tlv3502	

Design Alternate Op Amp

OPA2353	
V_{ss}	2.7V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	3mV
I_q	5.2mA
I_b	0.5pA
UGBW	44MHz
SR	22V/µs
#Channels	2
www.ti.com/product/opa2353	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

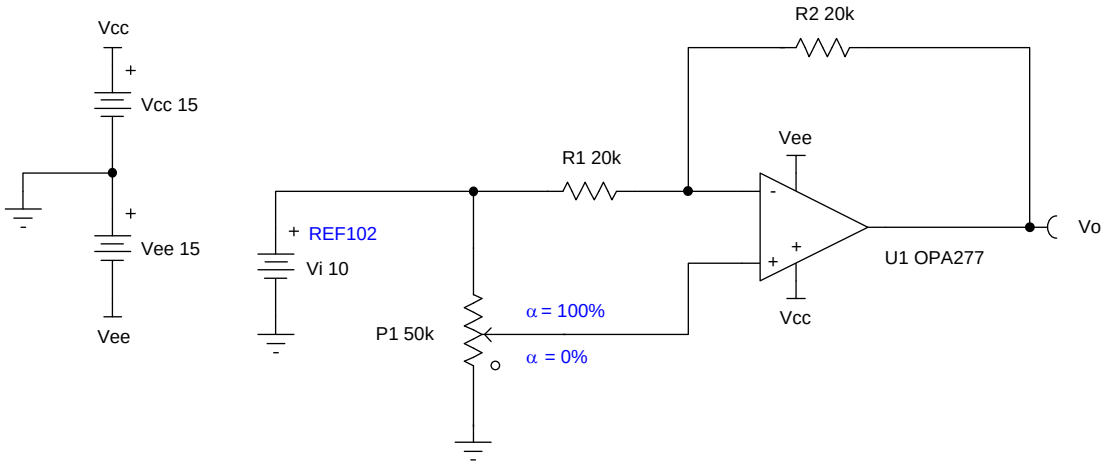
Adjustable reference voltage circuit

Design Goals

Input	Output		Supply	
V_i	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}
10V	–10V	10V	15V	–15V

Design Description

This circuit combines an inverting and non-inverting amplifier to make a reference voltage adjustable from the negative of the input voltage up to the input voltage. Gain can be added to increase the maximum negative reference level.

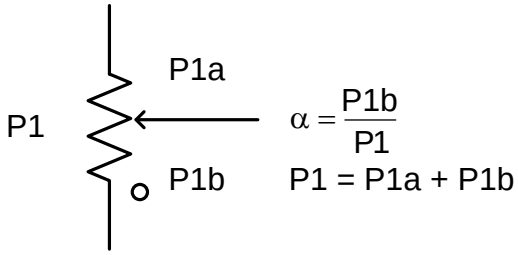


Design Notes

1. Observe the common-mode and output swing limitations of the op amp.
2. Mismatch in R_1 and R_2 results in a gain error. Selecting $R_2 > R_1$ increases the maximum negative voltage, and selecting $R_2 < R_1$ decreases the maximum negative voltage. In either case, the maximum positive voltage is always equal to the input voltage. This relationship is inverted if a negative input reference voltage is used.
3. Select the potentiometer based on the desired resolution of the reference. Generally, the potentiometers can be set accurately to within one-eighth of a turn. For a 10-turn pot this means alpha (∝) may be off by as much as 1.25%.

Design Steps

Alpha represents the potentiometer setting relative to ground. This is the fraction of the input voltage that will be applied to the non-inverting terminal of the op amp and amplified by the non-inverting gain.



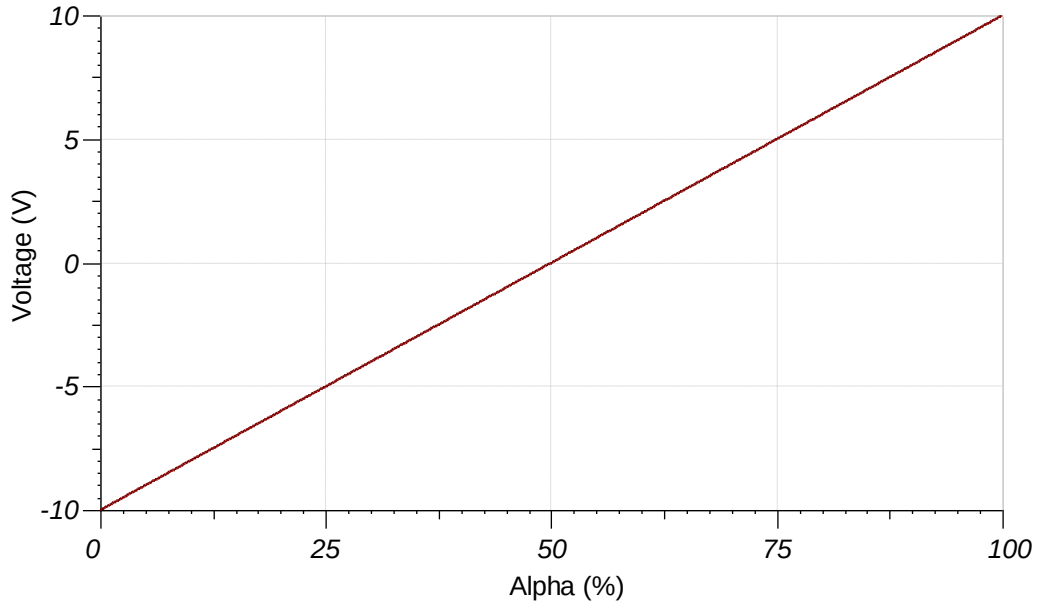
The transfer function of this circuit follows:

$$\frac{V_o}{V_i} = -\frac{R_2}{R_1} + \alpha\left(1 + \frac{R_2}{R_1}\right)$$

1. If $R_2 = R_1 = 20k\Omega$, then the equation for V_o simplifies as the following shows:
   $$V_o = (2\alpha - 1) \times V_i$$
2. If $V_i = 10V$ and $\propto = 0.75$, the value of V_o can be determined.
   $$V_o = (2 \times 0.75 - 1) \times 10 = 5V$$

Design Simulations

DC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the TINA-TI™ circuit simulation file, SBOMAU2.

See TI Precision Labs - Op Amps.

Design Featured Op Amp

OPA277	
V_{ss}	4V to 36V
V_{inCM}	V_{ee}+2V to V_{cc}–2V
V_{out}	V_{ee}+0.5V to V_{cc}–1.2V
V_{os}	10µV
I_q	790µA/Ch
I_b	500pA
UGBW	1MHz
SR	0.8V/µs
#Channels	1,2,4
http://www.ti.com/product/opa277	

Design Alternate Op Amp

OPA172	
V_{ss}	4.5V to 36V
V_{inCM}	V_{ee}–0.1V to V_{cc}–2V
V_{out}	Rail-to-rail
V_{os}	200µV
I_q	1.6mA/Ch
I_b	8pA
UGBW	10MHz
SR	10V/µs
#Channels	1,2,4
http://www.ti.com/product/opa172	

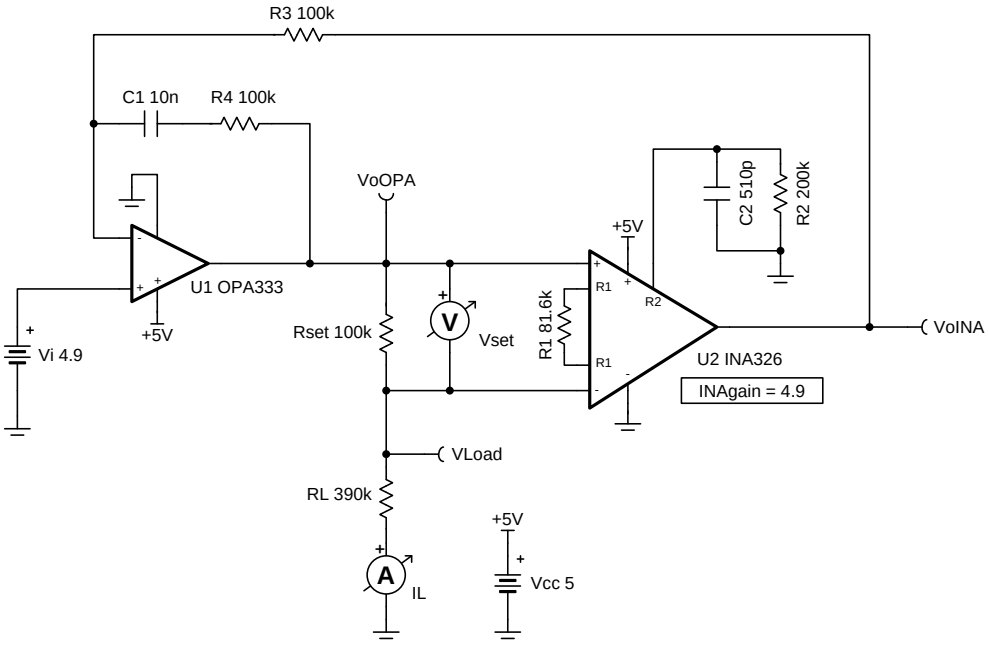
Low-level voltage-to-current converter circuit

Design Goals

Input		Output		Supply		Load Resistance (R_L)	
V_{iMin}	V_{iMax}	I_{LMin}	I_{LMax}	V_{cc}	V_{ee}	R_{LMin}	R_{LMax}
0.49V	4.9V	1µA	10µA	5V	0V	0Ω	390kΩ

Design Description

This circuit delivers a precise low-level current, I_L, to a load, R_L. The design operates on a single 5-V supply and uses one precision low-drift op amp and one instrumentation amplifier. Simple modifications can change the range and accuracy of the voltage-to-current (V-I) converter.



Design Notes

1. Voltage compliance is dominated by op amp linear output swing (see data sheet A_{OL} test conditions) and instrumentation amplifier linear output swing. See the Common-Mode Input Range Calculator for Instrumentation Amplifiers for more information.
2. Voltage compliance, along with R_{LMin}, R_{LMax}, and R_{set} bound the I_L range.
3. Check op amp and instrumentation amplifier input common-mode voltage range.
4. Stability analysis must be done to choose R_4 and C_1 for stable operation.
5. Loop stability analysis to select R_4 and C_1 will be different for each design. The compensation shown is only valid for the resistive load ranges used in this design. Other types of loads, op amps, or instrumentation amplifiers, or both will require different compensation. See the ***Design References*** section for more op amp stability resources.

Design Steps

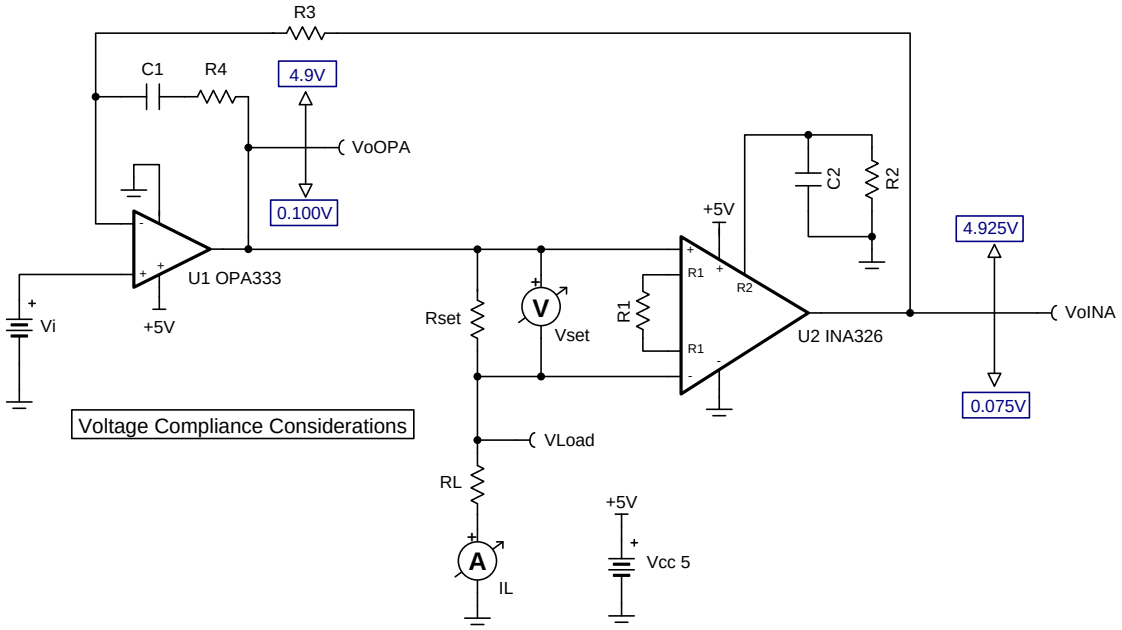
1. Select R_{set} and check I_{LMin} based on voltage compliance.

$$I_{LMax} = \frac{V_{oOPAMax}}{R_{set} + R_{LMax}}$$

$$10\mu A = \frac{4.9V}{R_{set} + 390k\Omega} \rightarrow R_{set} = 100k\Omega$$

$$I_{LMin} = \frac{V_{oOPAMin}}{R_{set} + R_{LMin}}$$

$$I_{LMin} = \frac{0.1V}{100k\Omega + 0\Omega} = 1\mu A$$



2. Compute instrumentation amplifier gain, G.

$$V_{setMin} = I_{LMin} \times R_{set} = 1\mu A \times 100k\Omega = 0.1V$$

$$V_{setMax} = I_{LMax} \times R_{set} = 10\mu A \times 100k\Omega = 1V$$

$$G = \frac{V_{iMax} - V_{iMin}}{V_{setMax} - V_{setMin}}$$

$$G = \frac{4.9V - 0.49V}{1V - 0.1V} = 4.9$$

3. Choose R_1 for INA326 instrumentation amplifier gain, G. Use data sheet recommended R_2 = 200kΩ and C_2 = 510pF.

$$G = 2 \times \left(\frac{R_2}{R_1}\right)$$

$$R_1 = \frac{2 \times R_2}{G}$$

$$R_1 = \left(\frac{2 \times 200k\Omega}{4.9}\right) = 81.6327k\Omega \approx 81.6k\Omega$$

4. The final transfer function of the circuit follows:

$$I_L = \frac{V_i}{G \times R_{set}}$$

$$I_L = \frac{V_i}{4.9 \times 100k\Omega} = \frac{V_i}{490k\Omega}$$

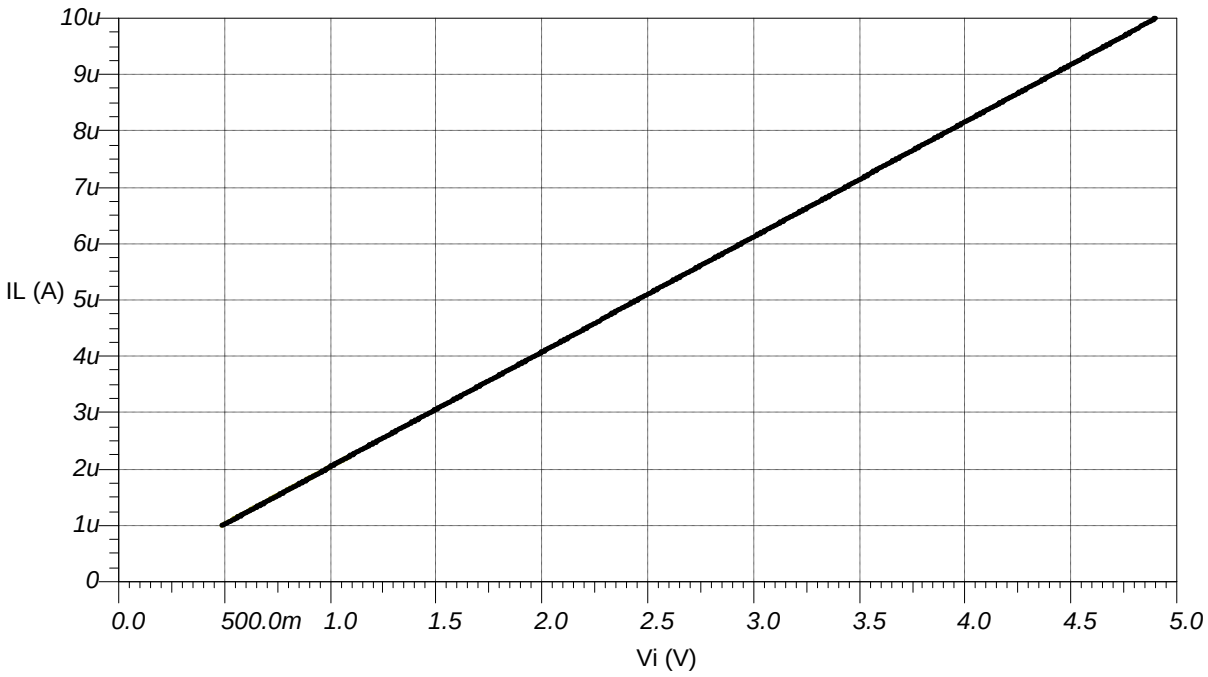
$$V_i = 0.49V \rightarrow I_L = 1\mu A$$

$$V_i = 4.9V \rightarrow I_L = 10\mu A$$

Design Simulations

DC Simulation Results

V_i	R_L	I_L	V_{oOPA}	V_{oOPA} Compliance	V_{oINA}	V_{oINA} Compliance
0.49V	0Ω	0.999627µA	99.982723mV	100mV to 4.9V	490.013346mV	75mV to 4.925V
0.49V	390kΩ	0.999627µA	489.837228mV	100mV to 4.9V	490.013233mV	75mV to 4.925V
4.9V	0Ω	9.996034µA	999.623352mV	100mV to 4.9V	4.900016V	75mV to 4.925V
4.9V	390kΩ	9.996031µA	4.898075V	100mV to 4.9V	4.900015V	75mV to 4.925V



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the TINA-TI™ circuit simulation file, SBOMAT8.

See TIPD107, http://www.ti.com/tool/TIPD107.

See Solving Op Amp Stability Issues - E2E FAQ.

See TI Precision Labs - Op Amps.

Design Featured Op Amp

OPA333	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	2µV
I_q	17µA/Ch
I_b	70pA
UGBW	350kHz
SR	0.16V/µs
#Channels	1,2
http://www.ti.com/product/opa333	

Design Featured Instrumentation Amplifier

INA326	
V_{ss}	2.7V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	20µV
I_q	2.4mA
I_b	0.2nA
UGBW	1kHz (set by 1kHz filter)
SR	0.012V/µs (set by 1kHz filter)
#Channels	1
http://www.ti.com/product/INA326	

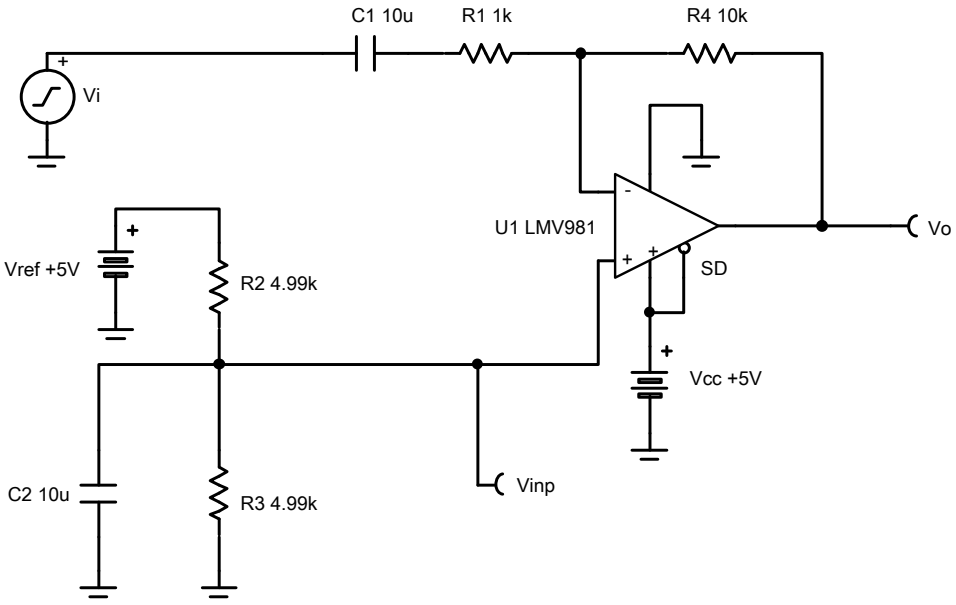
AC coupled (HPF) inverting amplifier circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–240mV	240mV	0.1V	4.9V	5V	0V	5V

Design Description

This circuit amplifies an AC signal and shifts the output signal so that it is centered at half the power supply voltage. Note that the input signal has zero DC offset so it swings above and below ground. The key benefit of this circuit is that it accepts signals which swing below ground even though the amplifier does not have a negative power supply.



Design Notes

1. R_1 sets the AC input impedance. R_4 loads the op amp output.
2. Use low feedback resistances to reduce noise and minimize stability concerns.
3. Set the output range based on linear output swing (see A_{ol} specification).
4. The cutoff frequency of the circuit is dependent on the gain bandwidth product (GBP) of the amplifier. Additional filtering can be accomplished by adding a capacitor in parallel to R_4. Adding a capacitor in parallel with R_4 will also improve stability of the circuit if high-value resistors are used.

Design Steps

1. Select R_1 and R_4 to set the AC voltage gain.

   $R_1 = 1\ \ k\Omega$ (Standard Value)

   $R_4 = R_1 \times |G_{ac}| = 1\ \ k\Omega \times |-10\frac{V}{V}| = 10k\Omega$ (Standard Value)

2. Select R_2 and R_3 to set the DC output voltage to 2.5V.

   $R_3 = 4.99k\Omega$ (Standard Value)

   $R_2 = \frac{R_3 \times V_{ref}}{V_{DC}} - R_3 = \frac{4.99k\Omega \times 5V}{2.5V} - 4.99k\Omega = 4.99k\Omega$

3. Choose a value for the lower cutoff frequency, f_l, then calculate C_1.

   $f_l = 16Hz$

   $C_1 = \frac{1}{2 \times \pi \times R_1 \times f_l} = \frac{1}{2 \times \pi \times 1\ \ k\Omega \times 16Hz} = 9.94\mu F \approx 10\mu F$ (Standard Value)

4. Choose a value for f_{div}, then calculate C_2.

   $f_{div} = 6.4Hz$

   $R_{div} = \frac{R_2 \times R_3}{R_2 + R_3} = \frac{4.99k\Omega \times 4.99k\Omega}{4.99k\Omega + 4.99k\Omega} = 2.495k\Omega$

   $C_2 = \frac{1}{2 \times \pi \times R_{div} \times f_{div}} = \frac{1}{2 \times \pi \times 2.495k\Omega \times 6.4Hz} = 9.96\mu F \approx 10\mu F$ (Standard Value)

5. The upper cutoff frequency, f_h, is set by the noise gain of this circuit and the gain bandwidth (GBW) of the device (LMV981).

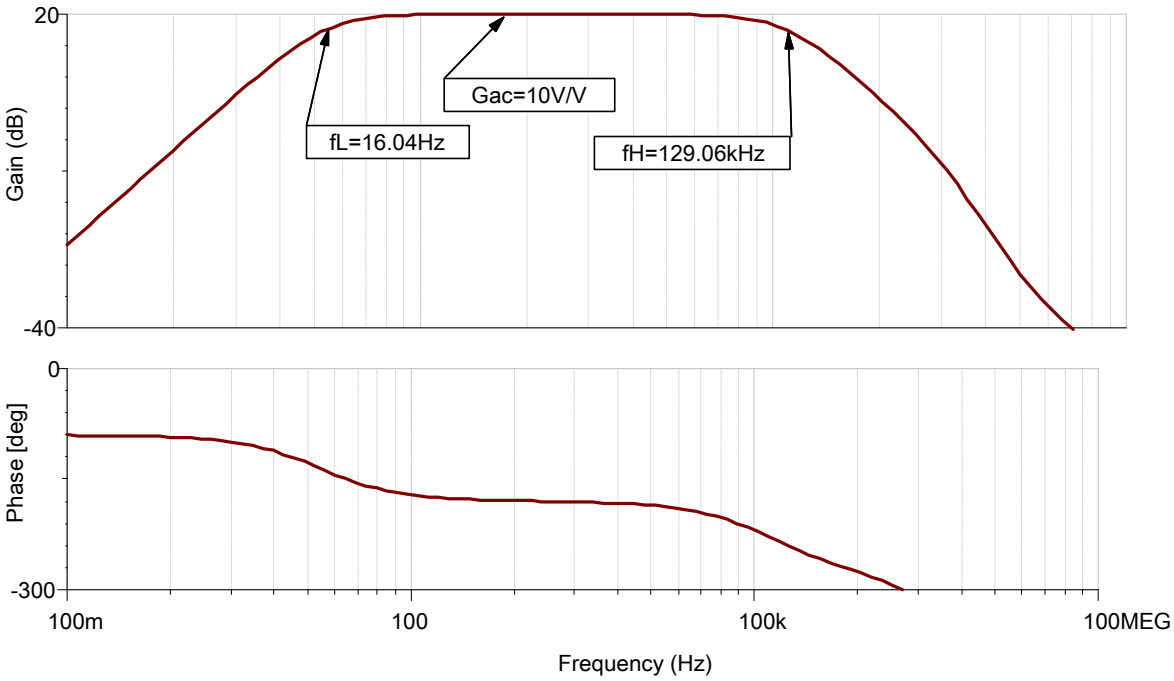
   $GBW = 1.5MHz$

   $G_{noise} = 1 + \frac{R_4}{R_1} = 1 + \frac{10k\Omega}{1\ \ k\Omega} = 11\frac{V}{V}$

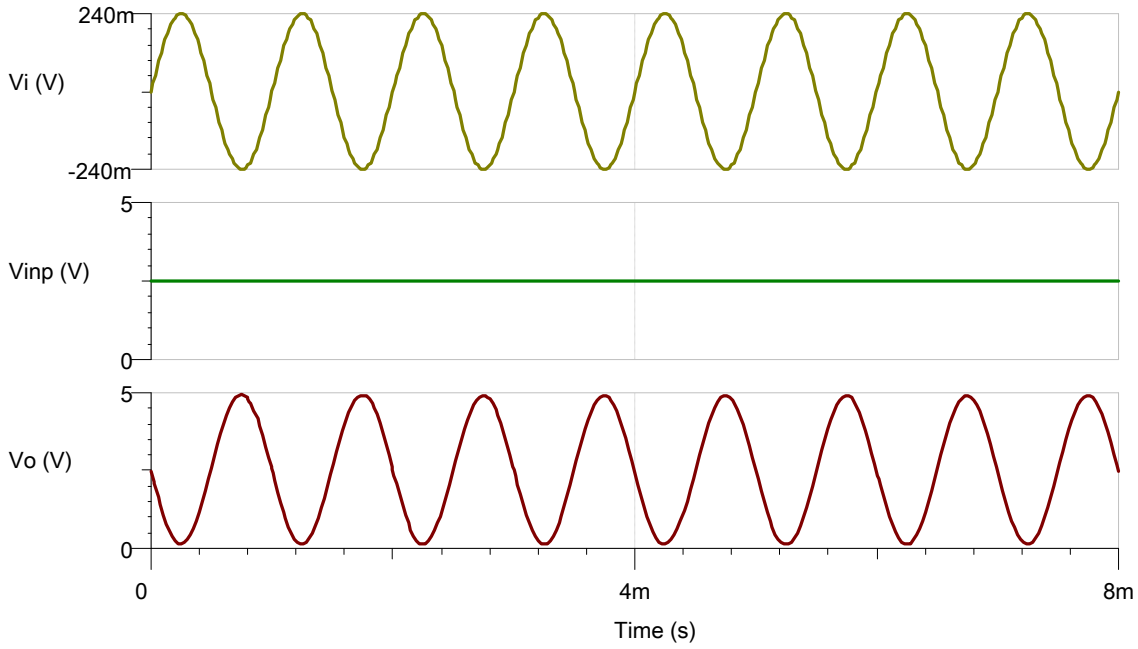
   $f_h = \frac{GBW}{G_{noise}} = \frac{1.5MHz}{11\frac{V}{V}} = 136.3kHz$

Design Simulations

AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC504.

See TIPD185, www.ti.com/tool/tipd185.

Design Featured Op Amp

LMV981	
V_{cc}	1.8V to 5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1mV
I_q	116µA
I_b	14nA
UGBW	1.5MHz
SR	0.42V/µs
#Channels	1, 2
www.ti.com/product/lmv981-n	

Design Alternate Op Amp

LMV771	
V_{cc}	2.7V to 5V
V_{inCM}	V_{ee} to (V_{cc}–0.9V)
V_{out}	Rail-to-rail
V_{os}	0.25mV
I_q	600µA
I_b	–0.23pA
UGBW	3.5MHz
SR	1.5V/µs
#Channels	1, 2
www.ti.com/product/lmv771	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

AC coupled (HPF) non-inverting amplifier circuit

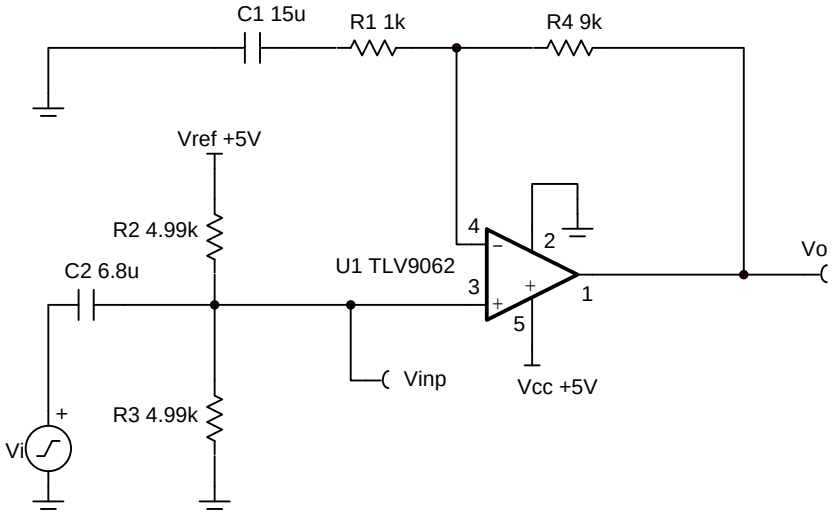
Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–240mV	240mV	0.1V	4.9V	5V	0V	5V

Lower Cutoff Freq. (f_L)	Upper Cutoff Freq. (f_H)	AC Gain (G_{ac})
16Hz	≥ 1MHz	10V/V

Design Description

This circuit amplifies an AC signal, and shifts the output signal so that it is centered at one-half the power supply voltage. Note that the input signal has zero DC offset so it swings above and below ground. The key benefit of this circuit is that it accepts signals which swing below ground even though the amplifier does not have a negative power supply.



Design Notes

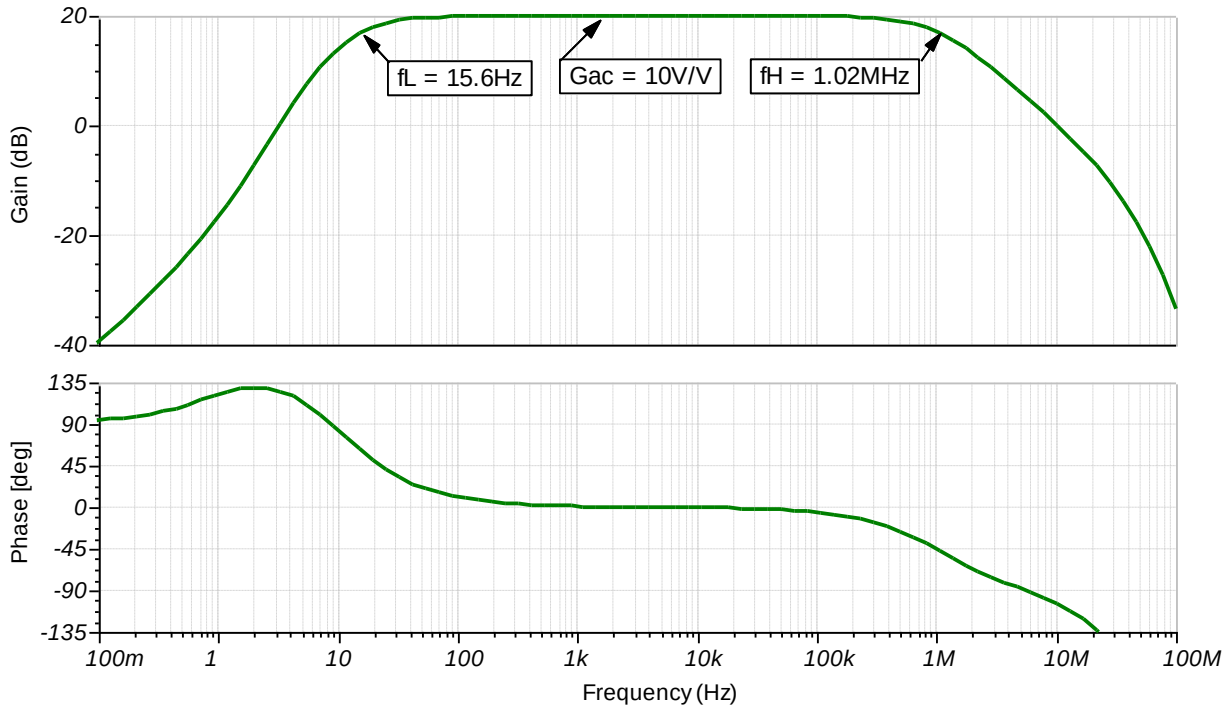
1. The voltage at V_{inp} sets the input common-mode voltage.
2. R_2 and R_3 load the input signal for AC frequencies.
3. Use low feedback resistance for low noise.
4. Set the output range based on linear output swing (see A_{ol} specification of op amp).
5. The circuit has two real poles that determine the high-pass filter –3dB frequency. Set them both to $f_L/1.557$ to achieve –3dB at the lower cutoff frequency (f_L).

Design Steps

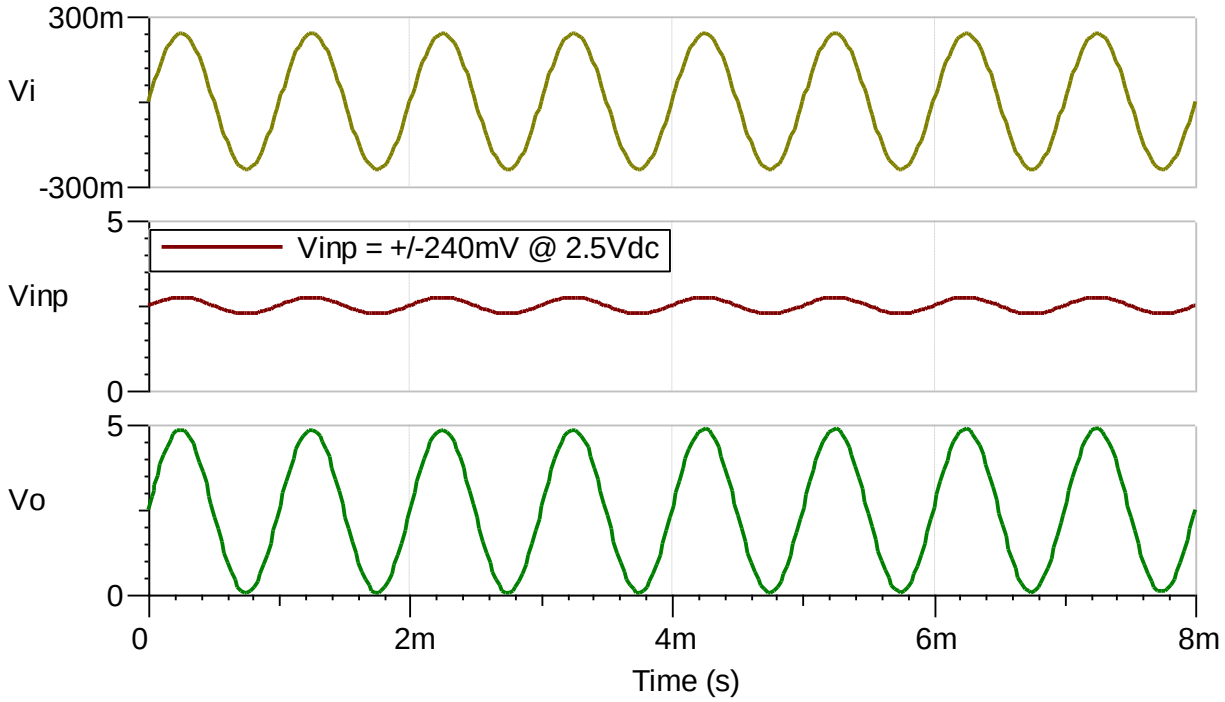
1. Select R_1 and R_4 to set the AC voltage gain.
   $R_1 = 1 \ \ k\Omega$ (Standard Value)
   $R_4 = R_1 \times (G_{ac} - 1) = 1 \ \ k\Omega \times (10\frac{V}{V} - 1) = 9k\Omega$ (Standard Value)
2. Select R_2 and R_3 to set the DC output voltage (V_{DC}) to 2.5V, or mid–supply.
   $R_3 = 4.99k\Omega$ (Standard Value)
   $R_2 = \frac{R_3 \times V_{ref}}{V_{DC}} - R_3 = \frac{4.99k\Omega \times 5V}{2.5V} - 4.99k\Omega = 4.99k\Omega$
3. Select C_1 based on f_L and R_1.
   $f_L = 16Hz$
   $C_1 = \frac{1}{2 \times \pi \times R_1 \times (\frac{f_L}{1.557})} = \frac{1}{2 \times \pi \times 1 \ \ k\Omega \times 10.3Hz} = 15.5\mu F \approx 15\mu F$ (Standard Value)
4. Select C_2 based on f_L, R_2, and R_3.
   $R_{div} = \frac{R_2 \times R_3}{R_2 + R_3} = \frac{4.99k\Omega \times 4.99k\Omega}{4.99k\Omega + 4.99k\Omega} = 2.495k\Omega$
   $C_2 = \frac{1}{2 \times \pi \times R_{div} \times (\frac{f_L}{1.557})} = \frac{1}{2 \times \pi \times 2.495k\Omega \times 10.3Hz} = 6.4\mu F \rightarrow 6.8\mu F(StandardValue)$
5. The upper cutoff frequency (f_H) is set by the non-inverting gain of this circuit and the gain bandwidth (GBW) of the device (TLV9062).
   $f_H = \frac{GBW \text{ of } TLV9062}{G_{ac}} = \frac{10MHz}{10\frac{V}{V}} = 1 \ \ MHz$

Design Simulations

AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC505.

See TIPD185, www.ti.com/tool/tipd185.

Design Featured Op Amp

TLV9062	
V_{cc}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	300µV
I_q	538µA
I_b	0.5pA
UGBW	10MHz
SR	6.5V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv9062	

Design Alternate Op Amp

OPA192	
V_{cc}	4.5V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	1mA/Ch
I_b	5pA
UGBW	10MHz
SR	20V/µs
#Channels	1, 2, 4
www.ti.com/product/opa192	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

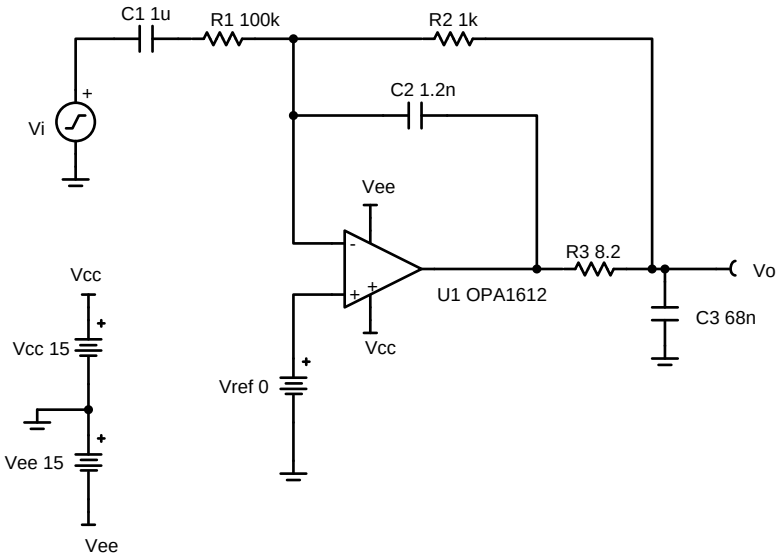
Band pass filtered inverting attenuator circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
$100mV_{pp}$	$50V_{pp}$	$1mV_{pp}$	$500mV_{pp}$	15V	–15V	0V

Design Description

This tunable band-pass attenuator reduces signal level by –40dB over the frequency range from 10Hz to 100kHz. It also allows for independent control of the DC output level. For this design, the pole frequencies were selected outside the pass band to minimize attenuation within the specified bandwidth range.



Design Notes

1. If a DC voltage is applied to V_{ref} be sure to check common mode limitations.
2. Keep R_3 as small as possible to avoid loading issues while maintaining stability.
3. Keep the frequency of the second pole in the low-pass filter (f_{p3}) at least twice the frequency of the first low-pass filter pole (f_{p2}).

Design Steps

1. Set the passband gain.

   $\text{Gain} = -\frac{R_2}{R_1} = -0.01\,\frac{V}{V}\ (-40\text{dB})$

   $R_1 = 100\text{k}\Omega$

   $R_2 = 0.01 \times R_1 = 1\ \ \text{k}\Omega$

2. Set high-pass filter pole frequency (f_{p1}) below f_l.

   $f_l = 10\text{Hz}, f_{p1} = 2.5\ \text{Hz}$

3. Set low-pass filter pole frequency (f_{p2} and f_{p3}) above f_h.

   $f_h = 100\text{kHz}$

   $f_{p2} = 150\text{kHz}$

   $f_{p3} \geq 2 \times f_{p2} = 300\text{kHz}$

   $f_{p3} = 300\text{kHz}$

4. Calculate C_1 to set the location of f_{p1}.

   $C_1 = \frac{1}{2\pi \times R_1 \times f_{p1}} = \frac{1}{2\pi \times 100\text{k}\Omega \times 2.5\text{Hz}} = 0.636\ \mu\text{F} \approx 1\ \ \mu\text{F}$ (Standard Value)

5. Select components to set f_{p2} and f_{p3}.

   $R_3 = 8.2\Omega$ (provides stability for cap loads up to 100nF)

   $C_2 = \frac{1}{2\pi \times (R_2 + R_3) \times f_{p2}} = \frac{1}{2\pi \times 1008.2\Omega \times 150\text{kHz}}$

   $= 1052\text{pF} \approx 1200\text{pF}$ (Standard Value)

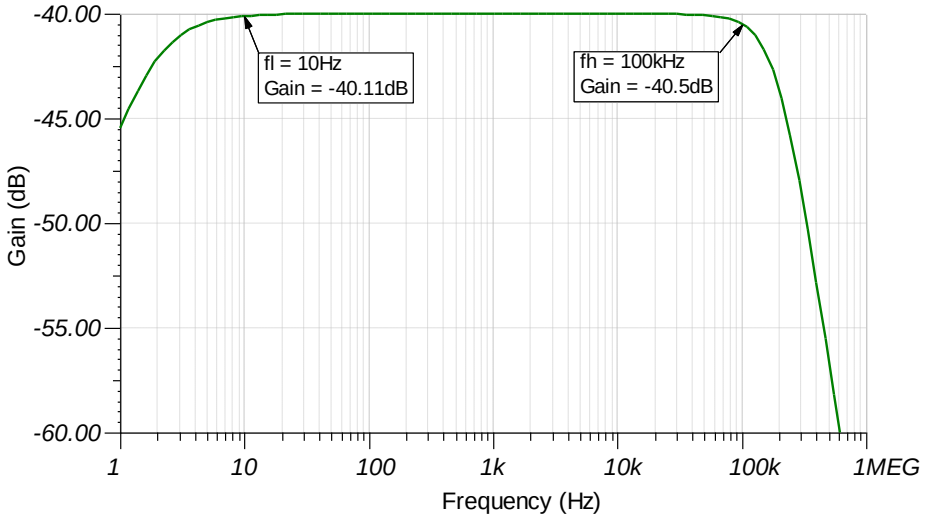
   $C_3 = \frac{1}{2\pi \times R_3 \times f_{p3}} = \frac{1}{2\pi \times 8.2\Omega \times 300\text{kHz}} = 64.7\ \text{nF} \approx 68\text{nF}$ (Standard Value)

Design Simulations

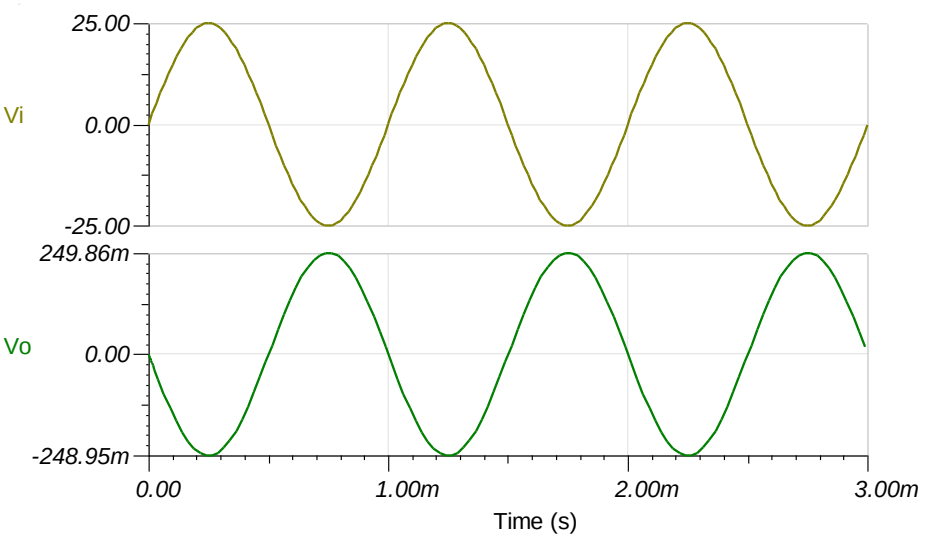
DC Simulation Results

The amplifier will pass DC voltages applied to the noninverting pin up to the common mode limitations of the op amp (±13V in this design)

AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC503.

See TIPD118, www.ti.com/tool/tipd118.

Design Featured Op Amp

OPA1612	
V_{ss}	4.5V to 36V
V_{inCM}	V_{ee}+2V to V_{cc}–2V
V_{out}	V_{ee}+0.2V to V_{cc}–0.2V
V_{os}	100µV
I_q	3.6mA/Ch
I_b	60nA
UGBW	40MHz
SR	27V/µs
#Channels	1, 2
www.ti.com/product/opa1612	

Design Alternate Op Amp

OPA172	
V_{ss}	4.5V to 36V
V_{inCM}	V_{ee}–100mV to V_{cc}–2V
V_{out}	Rail-to-rail
V_{os}	200µV
I_q	1.6mA/Ch
I_b	8pA
UGBW	10MHz
SR	10V/µs
#Channels	1, 2, 4
www.ti.com/product/opa172	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

Fast-settling low-pass filter circuit

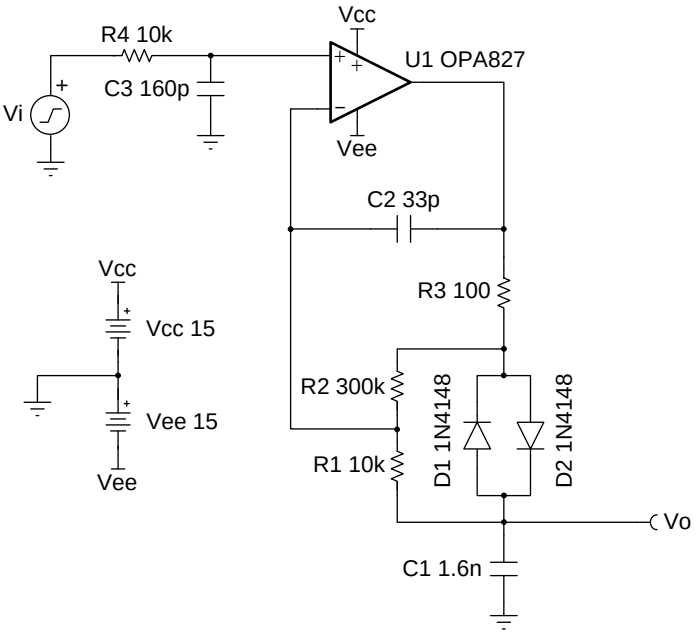
Design Goals

Input		Output		Supply	
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}
–12V	12V	–12V	12V	15V	–15V

Cutoff Frequency (f_c)	Diode Threshold Voltage (V_t)
10kHz	20mV

Design Description

This low-pass filter topology offers a significant improvement in settling time over the conventional single-pole RC filter. This is achieved through the use of diodes D_1 and D_2, that allow the filter capacitor to charge and discharge much faster when there is a large enough difference between the input and output voltages.



Design Notes

1. Observe the common-mode input limitations of the op amp.
2. Keeping C_1 small will ensure the op amp does not struggle to drive the capacitive load.
3. For the fastest settling time, use fast switching diodes.
4. The selected op amp should have sufficient output drive capability to charge C_1. R_3 limits the maximum charge current.

Design Steps

1. Select standard values for R_1 and C_1 based on f_C = 10kHz.

   $R_1 = 10k\Omega$

   $C_1 = \frac{1}{2\pi \times f_C \times R_1} = \frac{1}{2\pi \times 10kHz \times 10k\Omega} = 1.6nF$

2. Set the diode threshold voltage (V_t). This threshold is the minimum difference in voltage between the input and output that will result in diode conduction (fast capacitor charging and discharging).

   $V_t = \frac{V_f}{1+\frac{R_2}{R_1}} \approx \frac{0.6V}{1+\frac{R_2}{R_1}} = 20mV$

   $R_2 = (\frac{0.6V}{20mV} - 1) \times R_1 = 290k\Omega \approx 300k\Omega$ (standard 5% value)

3. Select components for noise pre-filtering.

   $f_{c2} = 10 \times f_c = 100kHz$

   $f_{c2} = \frac{1}{2\pi \times R_4 \times C_3}$

   Select $R_4 = R_1 = 10k\Omega$

   $C_3 = \frac{C_1}{10} = 160pF$

4. Add compensation components to stabilize U_1. R_3 limits the charge current into C_1 and also serves to isolate the capacitance from the op amp output when the diodes are conducting. Larger values will improve stability but increase C_1 charge time.

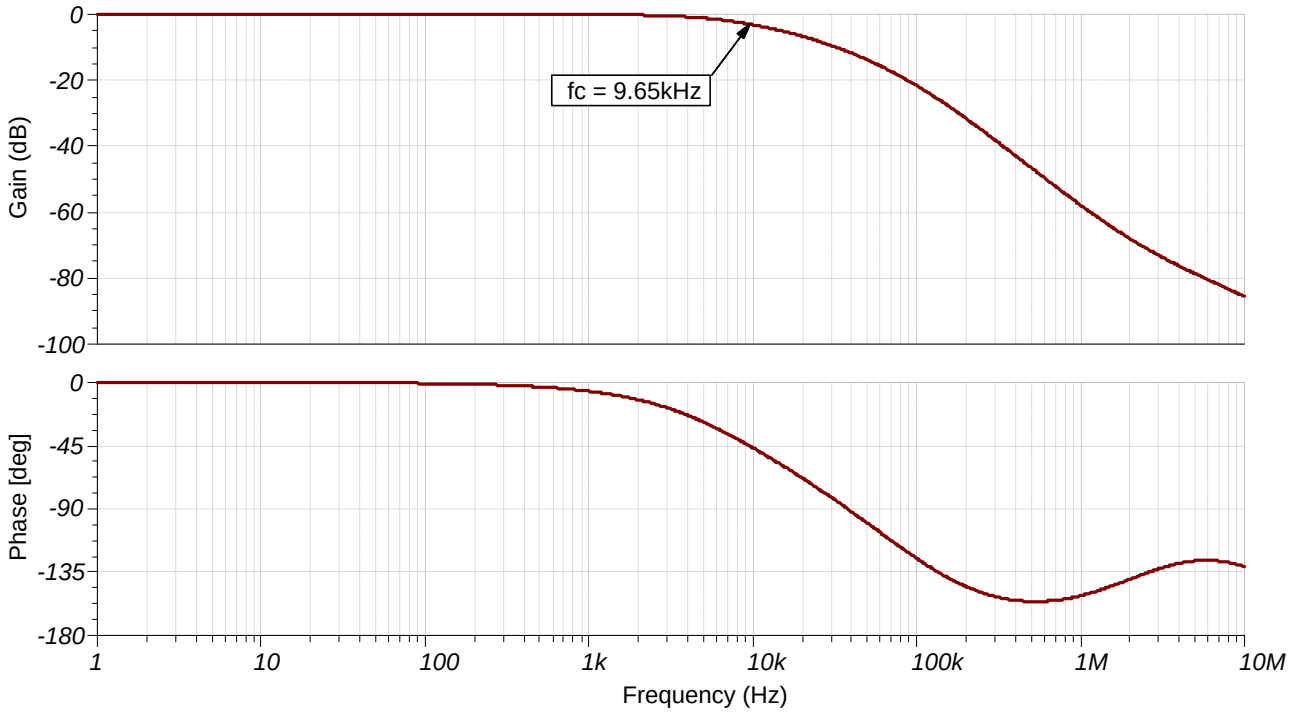
   Select $R_3 = 100\Omega$

5. C_2 provides local high frequency feedback to counteract the interaction between the input capacitance with the parallel combination of R_1 and R_2. To prevent interaction with C_1, select C_2 as the following shows:

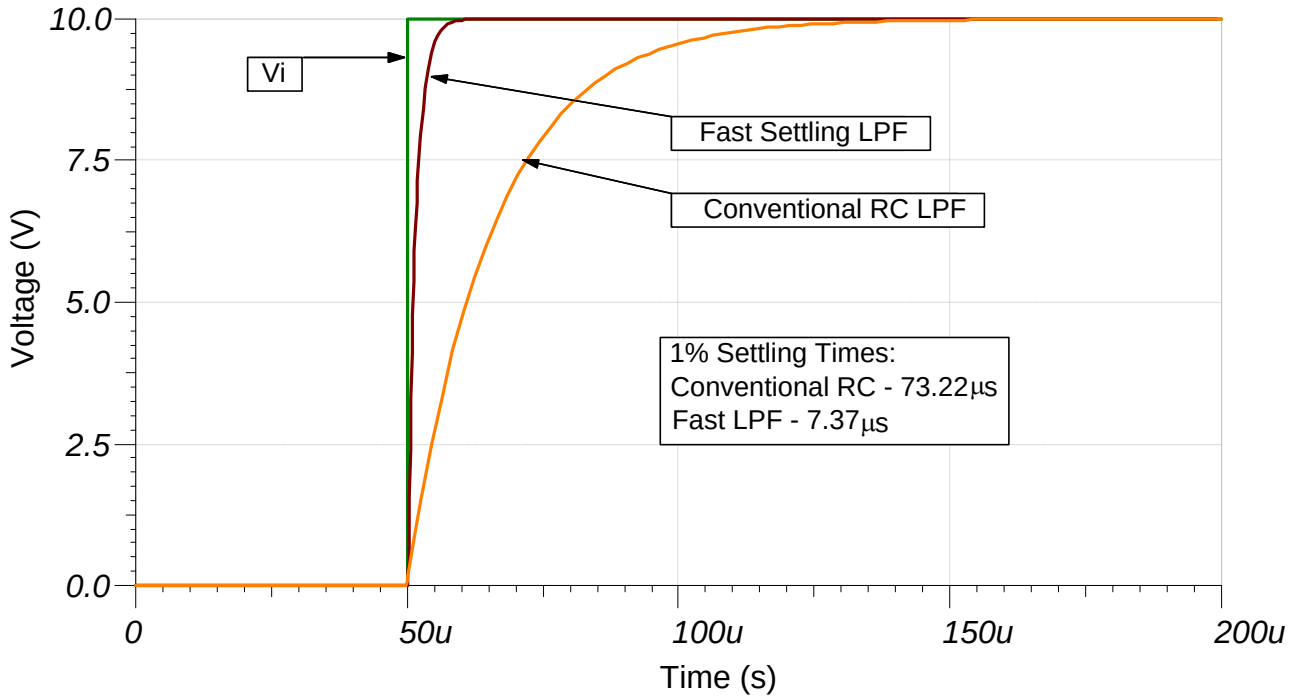
   Select $C_2 = \frac{C_1}{50} = 32pF \approx 33pF$ (standard value)

Design Simulations

AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See TINA-TI™ circuit simulation file, SBOMAU1.

For more information on many op amp topics including common-mode range, output swing, bandwidth, and how to drive an ADC, see TI Precision Labs.

Design Featured Op Amp

OPA827	
V_{ss}	8V to 36V
V_{inCM}	V_{ee}+3V to V_{cc}–3V
V_{out}	V_{ee}+3V to V_{cc}–3V
V_{os}	75µV
I_q	4.8mA
I_b	3pA
UGBW	22MHz
SR	28V/µs
#Channels	1
http://www.ti.com/product/opa827	

Design Alternate Op Amp

TLC072	
V_{ss}	4.5V to 16V
V_{inCM}	V_{ee}+0.5V to V_{cc}–0.8V
V_{out}	V_{ee}+350mV to V_{cc}–1V
V_{os}	390µV
I_q	2.1mA/Ch
I_b	1.5pA
UGBW	10MHz
SR	16V/µs
#Channels	1,2,4
http://www.ti.com/product/tlc072	

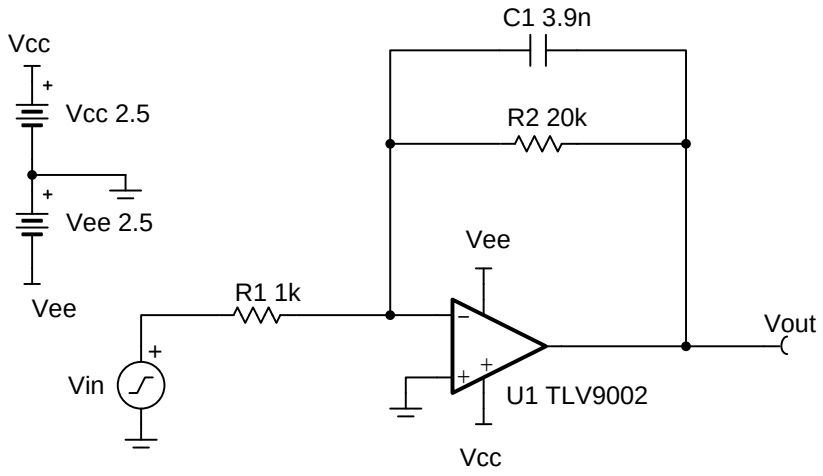
Low–pass filtered, inverting amplifier circuit

Design Goals

Input		Output		BW	Supply	
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	f_p	V_{ee}	V_{cc}
–0.1V	0.1V	–2V	2V	2kHz	–2.5V	2.5V

Design Description

This tunable low–pass inverting amplifier circuit amplifies the signal level by 26dB or 20V/V. R_2 and C_1 set the cutoff frequency for this circuit. The frequency response of this circuit is the same as that of a passive RC filter, except that the output is amplified by the pass–band gain of the amplifier. Low–pass filters are often used in audio signal chains and are sometimes called bass–boost filters.



Design Notes

1. C_1 and R_2 set the low–pass filter cutoff frequency
2. The common-mode voltage is set by the non–inverting input of the op amp, which in this case is mid–supply.
3. Using high value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.
4. R_2 and R_1 set the gain of the circuit.
5. The pole frequency f_p of 2kHz is selected for an audio bass–boost application.
6. Avoid placing capacitive loads directly on the output of the amplifier to minimize stability issues.
7. Large signal performance may be limited by slew rate. Therefore, check the maximum output swing versus frequency plot in the data sheet to minimize slew–induced distortion.
8. For more information on op amp linear operation region, stability, slew–induced distortion, capacitive load drive, driving ADCs and bandwidth please see the design references section.

Design Steps

The DC transfer function of this circuit is given below.

$$V_o = V_i \times \left(-\frac{R_2}{R_1}\right)$$

1. Pick resistor values for given passband gain.

$$\text{Gain} = \frac{R_2}{R_1} = 20\ \frac{V}{V}\ (26\ \text{dB})$$

$$R_1 = 1\ \text{k}\Omega$$

$$R_2 = \text{Gain} \times (R_1) = 20\ \frac{V}{V} \times 1\ \text{k}\Omega = 20\ \text{k}\Omega$$

2. Select low–pass filter pole frequency f_p

$$f_p = 2\ \text{kHz}$$

3. Calculate C_1 using R_2 to set the location of f_p.

$$f_p = \frac{1}{2\pi \times R_2 \times C_1} = 2\ \text{kHz}$$

$$C_1 = \frac{1}{2\pi \times R_2 \times f_p} = \frac{1}{2\pi \times 20\ \text{k}\Omega \times 2\ \text{kHz}} = 3.98\text{nF} \approx 3.9\text{nF}\ (\text{Standard Value})$$

4. Calculate the minimum slew rate required to minimize slew–induced distortion.

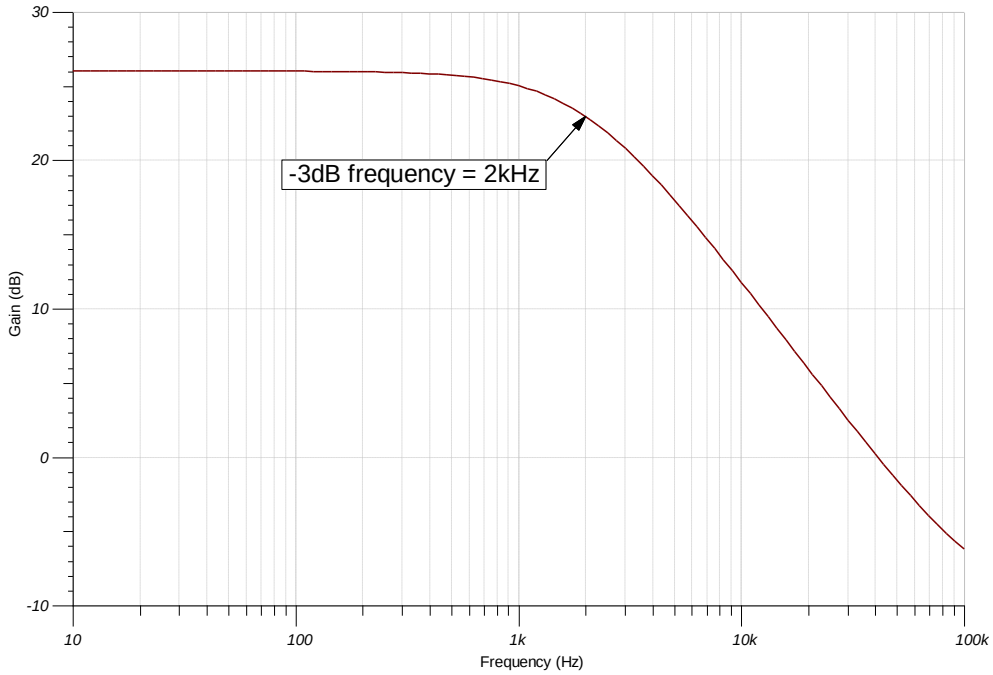
$$V_p = \frac{SR}{2 \times \pi \times f} \rightarrow SR > 2 \times \pi \times f \times V_p$$

$$SR > 2 \times \pi \times 2\ \text{kHz} \times 2\ \text{V} = 0.25\ \frac{V}{\mu s}$$

5. $SR_{TLV9002}$ = 2V/µs, therefore it meets this requirement

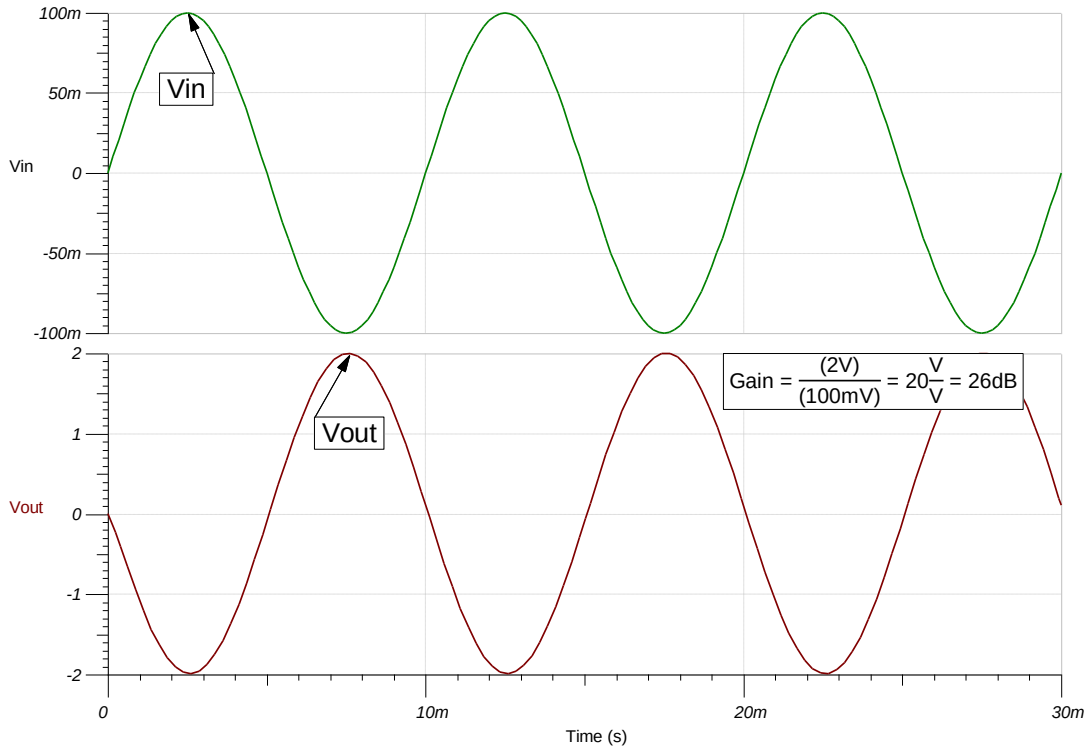
Design Simulations

AC Simulation Results

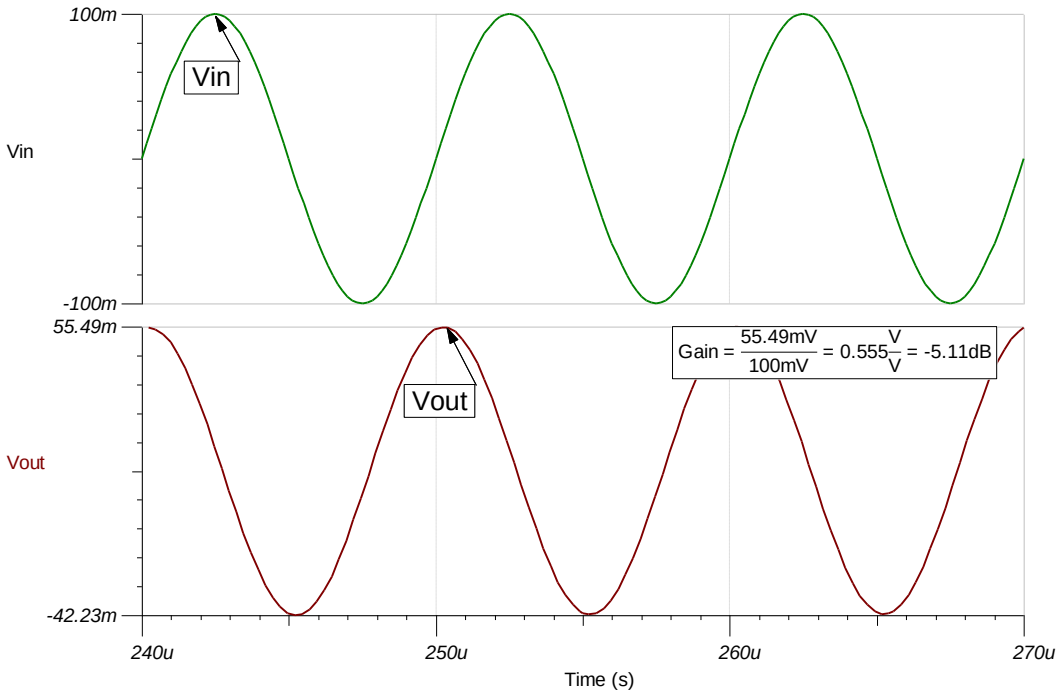


Transient Simulation Results

A 100 Hz, 0.2 V_{pp} sine wave yields a 4 V_{pp} output sine wave.



A 100 kHz, 0.2 V_{pp} sine wave yields a 0.1 V_{pp} output sine wave.



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOC523
3. TI Precision Designs TIPD185
4. TI Precision Labs

Design Featured Op Amp

TLV9002	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	0.4mV
I_q	60µA
I_b	5pA
UGBW	1MHz
SR	2V/µs
#Channels	1,2,4
www.ti.com/product/tlv9002	

Design Alternate Op Amp

OPA375	
V_{ss}	2.25V to 5.5V
V_{inCM}	V_{ee} to V_{cc} –1.2V
V_{out}	Rail–to–rail
V_{os}	0.15mV
I_q	890µA
I_b	10pA
UGBW	10MHz
SR	4.75V/µs
#Channels	1
www.ti.com/product/opa375	

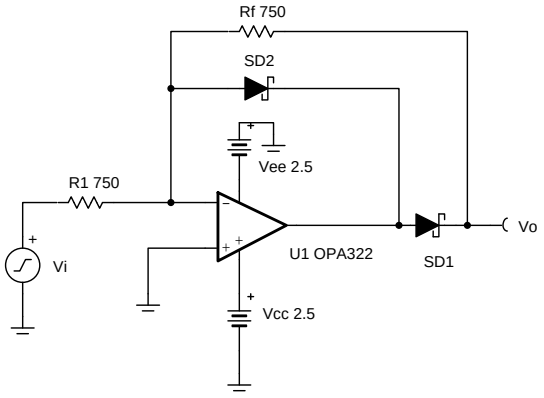
Half-wave rectifier circuit

Design Goals

Input		Output		Supply	
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}
±0.2mV$_{pp}$	±4V$_{pp}$	0.1V$_{p}$	2V$_{p}$	2.5V	–2.5V

Design Description

The precision half-wave rectifier inverts and transfers only the negative-half input of a time varying input signal (preferably sinusoidal) to its output. By appropriately selecting the feedback resistor values, different gains can be achieved. Precision half-wave rectifiers are commonly used with other op amp circuits such as a peak-detector or bandwidth limited non-inverting amplifier to produce a DC output voltage. This configuration has been designed to work for sinusoidal input signals between 0.2mV$_{pp}$ and 4V$_{pp}$ at frequencies up to 50kHz.



Design Notes

1. Select an op amp with a high slew rate. When the input signal changes polarities, the amplifier output must slew two diode voltage drops.
2. Set output range based on linear output swing (see A_{ol} specification).
3. Use fast switching diodes. High-frequency input signals will be distorted depending on the speed by which the diodes can transition from blocking to forward conducting mode. Schottky diodes might be a preferable choice, since these have faster transitions than pn-junction diodes at the expense of higher reverse leakage.
4. The resistor tolerance sets the circuit gain error.
5. Minimize noise errors by selecting low-value resistors.

Design Steps

1. Set the desired gain of the half-wave rectifier to select the feedback resistors.

$$V_o = \text{Gain} \times V_i$$

$$\text{Gain} = -\frac{R_f}{R_1} = -1$$

$$R_f = R_1 = 2 \times R_{eq}$$

   - Where R_{eq} is the parallel combination of R_1 and R_f

2. Select the resistors such that the resistor noise is negligible compared to the voltage broadband noise of the op amp.

$$E_{nr} = \sqrt{4 \times k_b \times T \times R_{eq}}$$

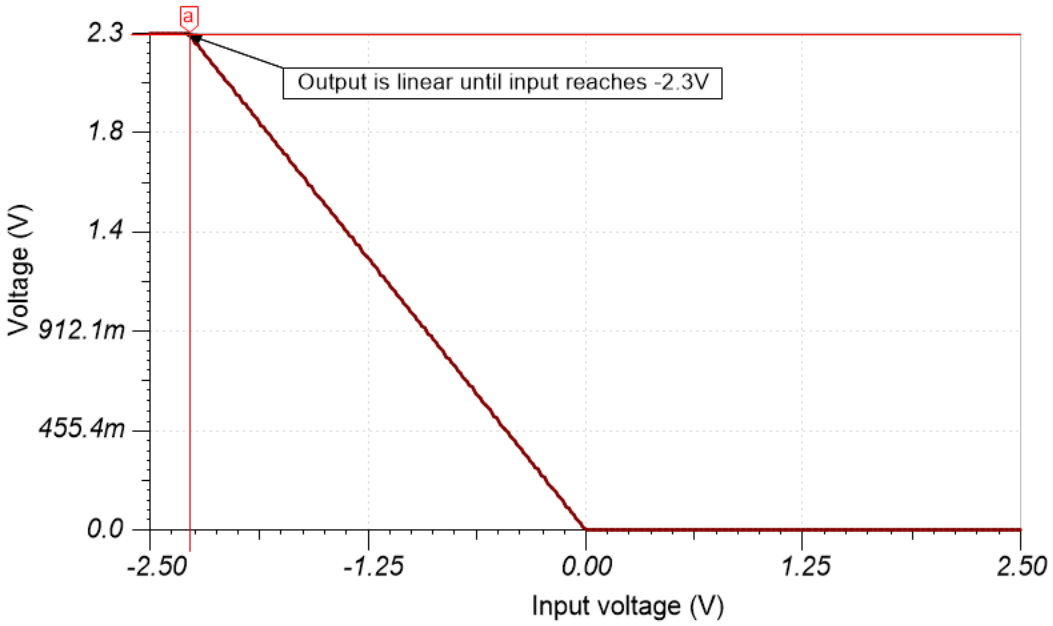
$$R_{eq} \le \frac{E_{nbb}^2}{4 \times k_b \times T \times 3^2} = (\text{Enbb})$$

$$= 7.5 \frac{nV}{\sqrt{Hz}} = \frac{(7.5 \times 10^{-9})^2}{4 \times 1.381 \times 10^{-23} \times 298 \times 3^2} = 380\Omega$$

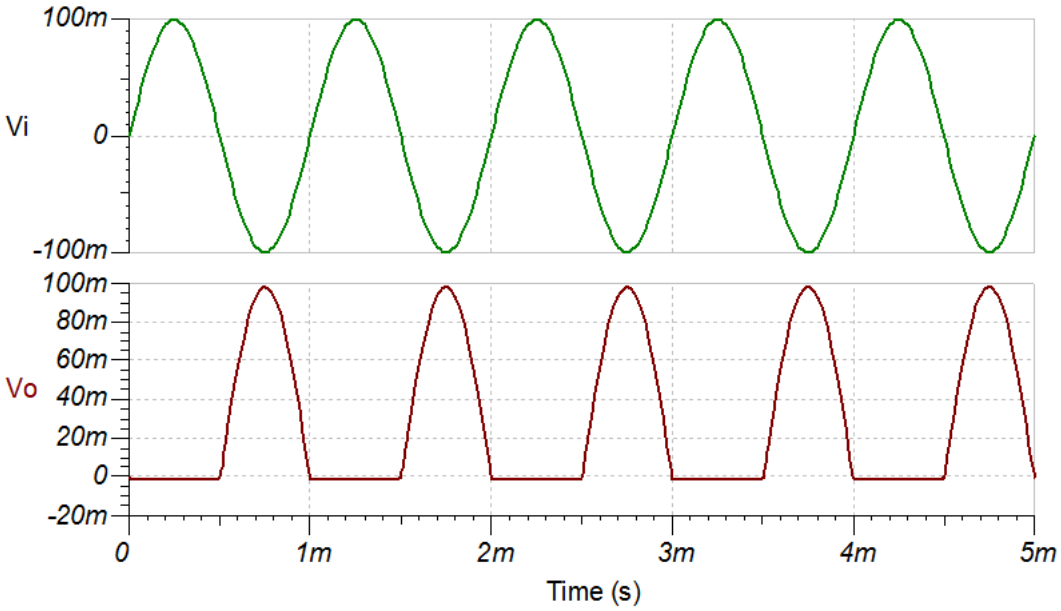
$$R_f = R_1 \le 760\Omega \rightarrow 750\Omega \text{ (Standard Value)}$$

Design Simulations

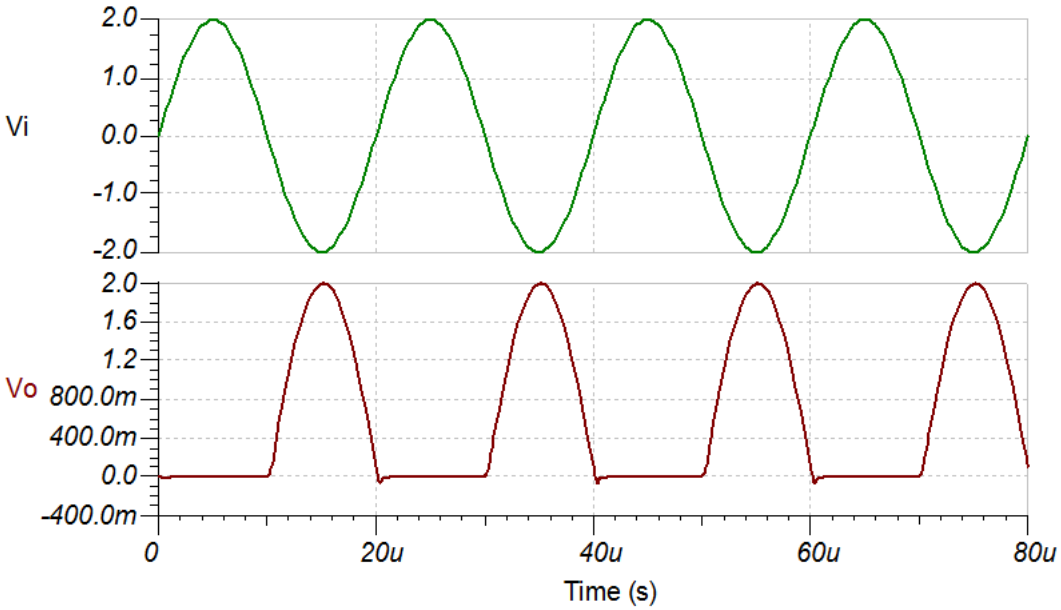
DC Simulation Results



Transient Simulation Results



200mV$_{pp}$ at 1kHz



2V$_{pp}$ at 50kHz

Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC509.

Design Featured Op Amp

OPA322	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	500µV
I_q	1.6mA/Ch
I_b	0.2pA
UGBW	20MHz
SR	10V/µs
#Channels	1, 2, 4
www.ti.com/product/opa322	

Design Alternate Op Amp

OPA2325	
V_{ss}	2.2V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	40µV
I_q	0.65mA/Ch
I_b	0.2pA
UGBW	10MHz
SR	5V/µs
#Channels	2µ
www.ti.com/product/opa2325	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page and link to Spice simulation file.

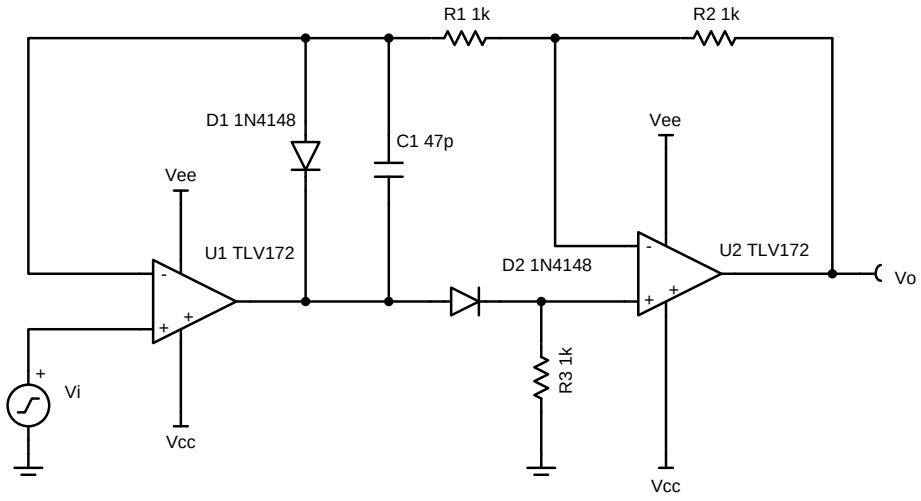
Full-wave rectifier circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
±25mV	±10V	25mV	10V	15V	–15V	0V

Design Description

This absolute value circuit can turn alternating current (AC) signals to single polarity signals. This circuit functions with limited distortion for ±10-V input signals at frequencies up to 50kHz and for signals as small as ±25mV at frequencies up to 1kHz.



Design Notes

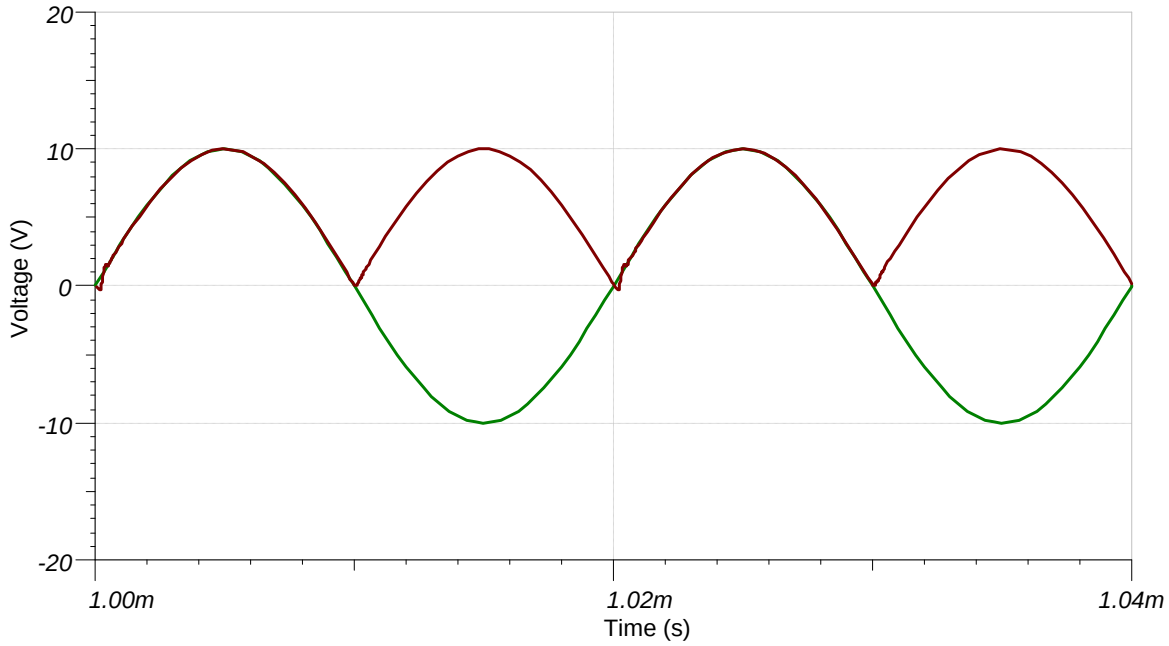
1. Be sure to select an op amp with sufficient bandwidth and a high slew rate.
2. For greater precision look for an op amp with low offset voltage, low noise, and low total harmonic distortion (THD).
3. The resistors were selected to be 0.1% tolerance to reduce gain error.
4. Selecting too large of a capacitor C_1 will cause large distortion on the transition edges when the input signal changes polarity. C_1 may not be required for all op amps.
5. Use a fast switching diode.

Design Steps

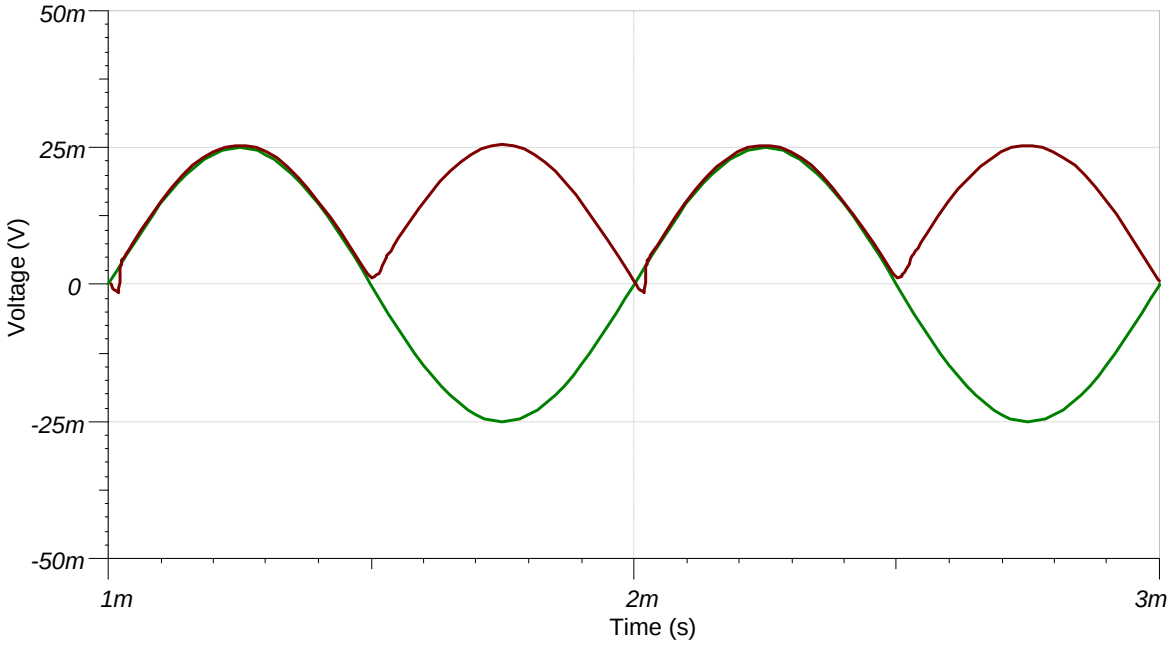
1. Select gain resistors.
   a. Gain for positive input signals.
      $\frac{V_o}{V_i} = 1\frac{V}{V}$
   b. Gain for negative input signals.
      $\frac{V_o}{V_i} = -\frac{R_2}{R_1} = -1\frac{V}{V}$
2. Select R_1 and R_2 to reduce thermal noise and to minimize voltage drops due to the reverse leakage current of the diode. These resistors will appear as loads to U_1 and U_2 during negative input signals.
   $R_1 = R_2 = 1 \quad k\Omega$
3. R_3 biases the non-inverting node of U_2 to GND during negative input signals. Select R_3 to be the same value as R_1 and R_2. U_1 must be able to drive the R_3 load during positive input signals.
   $R_3 = 1 \quad k\Omega$
4. Select C_1 based on the desired transient response. See the Design Reference section for more information.
   $C_1 = 47pF$

Design Simulations

Transient Simulation Results



±10V at 50-kHz Input



±25mV at 1-kHz Input

Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC517.

See TIPD139, www.ti.com/tool/tipd139.

Design Featured Op Amp

TLV172	
V_{cc}	4.5V to 36V
V_{inCM}	Vee to (Vcc–2V)
V_{out}	Rail-to-rail
V_{os}	0.5mV
I_q	1.6mA/Ch
I_b	10pA
UGBW	10MHz
SR	10V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv172	

Design Alternate Op Amp

OPA197	
V_{cc}	4.5V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	25µV
I_q	1mA/Ch
I_b	5pA
UGBW	10MHz
SR	20V/µs
#Channels	1, 2, 4
www.ti.com/product/opa197	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page and Spice simulation file.

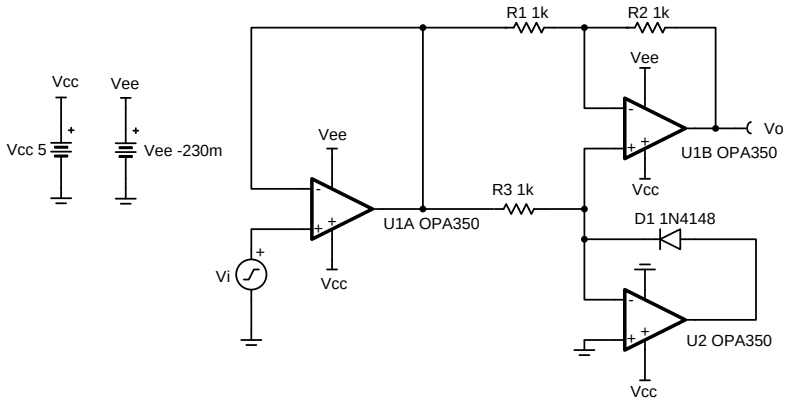
Single-supply, low-input voltage, full-wave rectifier circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
5mVpp	400mVpp	2.5mVpp	200mVpp	5V	–0.23V	0V

Design Description

This single-supply precision absolute value circuit is optimized for low-input voltages. It is designed to function up to 50kHz and has excellent linearity at signal levels as low as 5mVpp. The design uses a negative charge pump (such as LM7705) on the negative op amp supply rails to maintain linearity with signal levels near 0V.

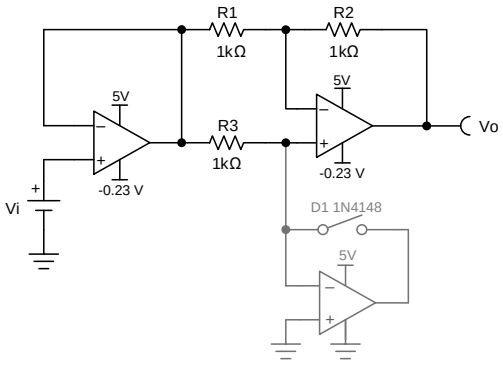


Design Notes

1. Observe common-mode and output swing limitations of op amps.
2. R_3 should be sized small enough that the leakage current from D_1 does not cause errors in positive input cycles while ensuring the op amp can drive the load.
3. Use a fast switching diode for D_1.
4. Removing the input buffer will allow for input signals with peak-to-peak values twice as large as the supply voltage at the expense of lower input impedance and slight gain error.
5. Use precision resistors to minimize gain error.

Design Steps

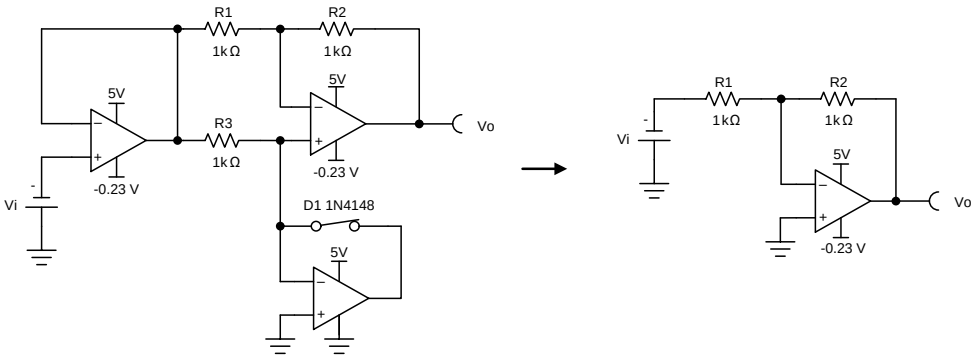
1. Circuit analysis for positive input signals.



$$\frac{V_o}{V_i} = \left(-\frac{R_2}{R_1}\right) + \left(1 + \frac{R_2}{R_1}\right) = 1$$

$$V_o = V_i$$

2. Circuit analysis for negative input signals.



$$\frac{V_o}{V_i} = \left(-\frac{R_2}{R_1}\right) = -1$$

$$V_o = -V_i$$

3. Select R_1, R_2, and R_3.

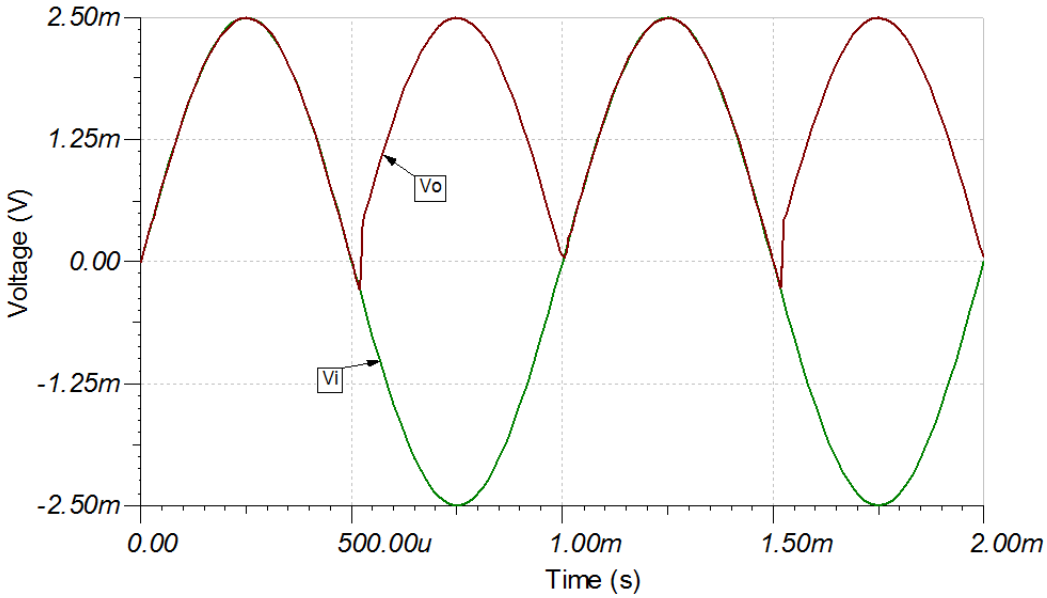
$$\frac{V_o}{V_i} = -\frac{R_2}{R_1}$$

If $R_2 = R_1$ then $V_o = -V_i$

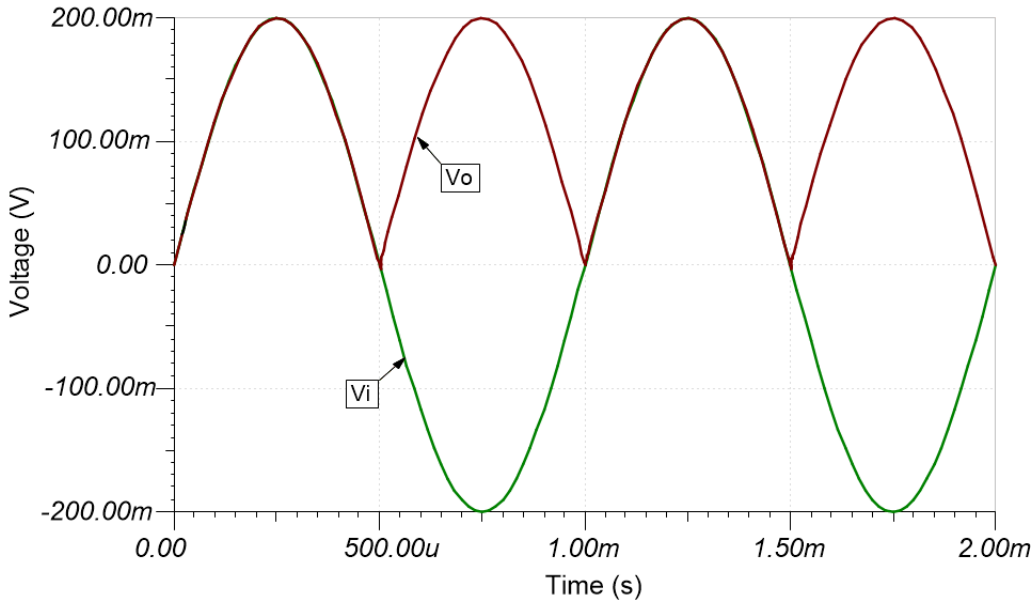
Set $R_1 = R_2 = R_3 = 1$ kΩ

Design Simulations

Transient Simulation Results



5mVpp at 1-kHz Input



400mVpp at 1-kHz Input

Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC506.

See TIPD124, www.ti.com/tool/tipd124.

Design Featured Op Amp

OPA350	
V_{ss}	2.7V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	150µV
I_q	5.2mA/Ch
I_b	0.5pA
UGBW	38MHz
SR	22V/µs
#Channels	1, 2, 4
www.ti.com/product/opa350	

Design Alternate Op Amp

OPA353	
V_{ss}	2.7V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	3mV
I_q	5.2mA
I_b	0.5pA
UGBW	44MHz
SR	22V/µs
#Channels	1, 2, 4
www.ti.com/product/opa353	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

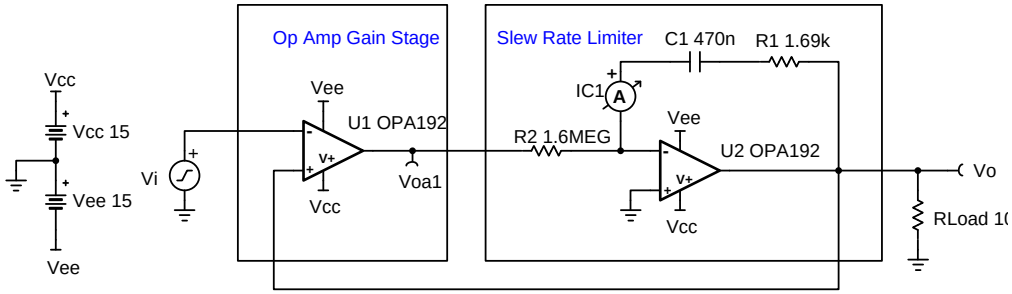
Slew rate limiter circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–10V	10V	–10V	10V	15V	–15V	0V

Design Description

This circuit controls the slew rate of an analog gain stage. This circuit is intended for symmetrical slew rate applications. The desired slew rate must be slower than that of the op amp chosen to implement the slew rate limiter.



Design Notes

1. The gain stage op-amp and slew rate limiting op amp should both be checked for stability.
2. Verify that the current demands for charging or discharging C_1 plus any load current out of U_2 will not limit the voltage swing of U_2.

Design Steps

1. Set slew rate and choose a standard value for the feedback capacitor, C_1.

   $C_1 = 470nF$

   $SR = 20\frac{V}{s}$

2. Choose the value of R_2 to set the capacitor current necessary for the desired slew rate.

   $SR = \frac{I_{C_1}}{C_1}$

   $20\frac{V}{s} = \frac{I_{C_1}}{470nF}$ where $I_{C_1} = 9.4\ \mu A$

   Gain stage op amp $V_{sat} = \pm 14.995$ (typical)

   $I_{C_1} = \frac{V_{sat}}{R_2}$

   $9.4\ \mu A = \frac{14.995V}{R_2}$, so $R_2 = 1.595\ M\Omega \approx 1.6M\Omega$ (Standard Value)

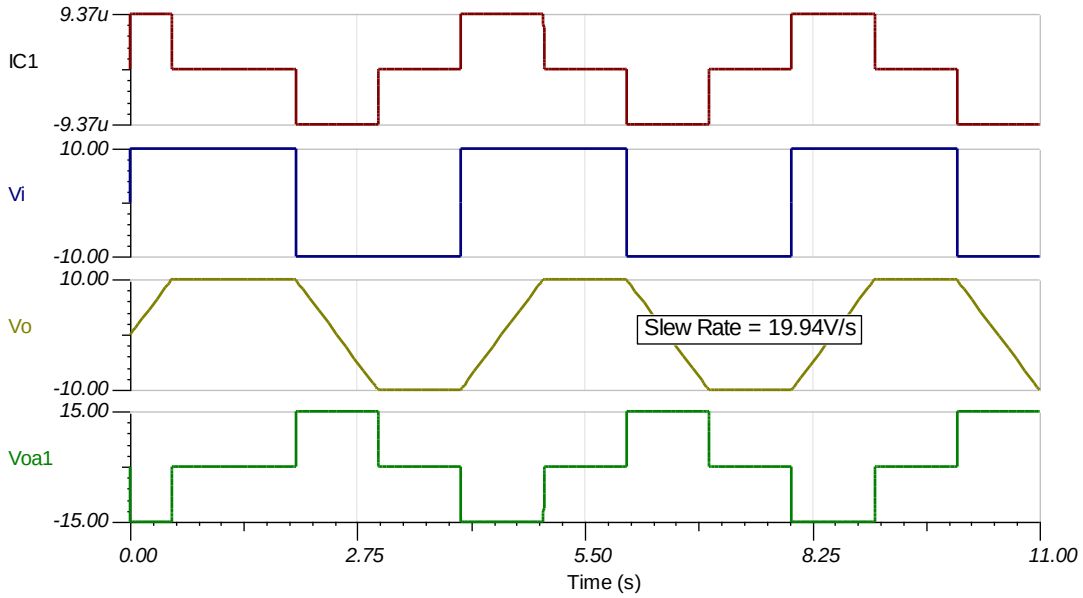
3. Compensate feedback network for stability. R_1 adds a pole to the $1/\beta$ network. This pole should be placed so that the $1/\beta$ curve levels off a decade before it intersects the open loop gain curve (200Hz, for this example).

   $f_p = \frac{1}{2\pi \times R_1 \times C_1} = 200Hz$

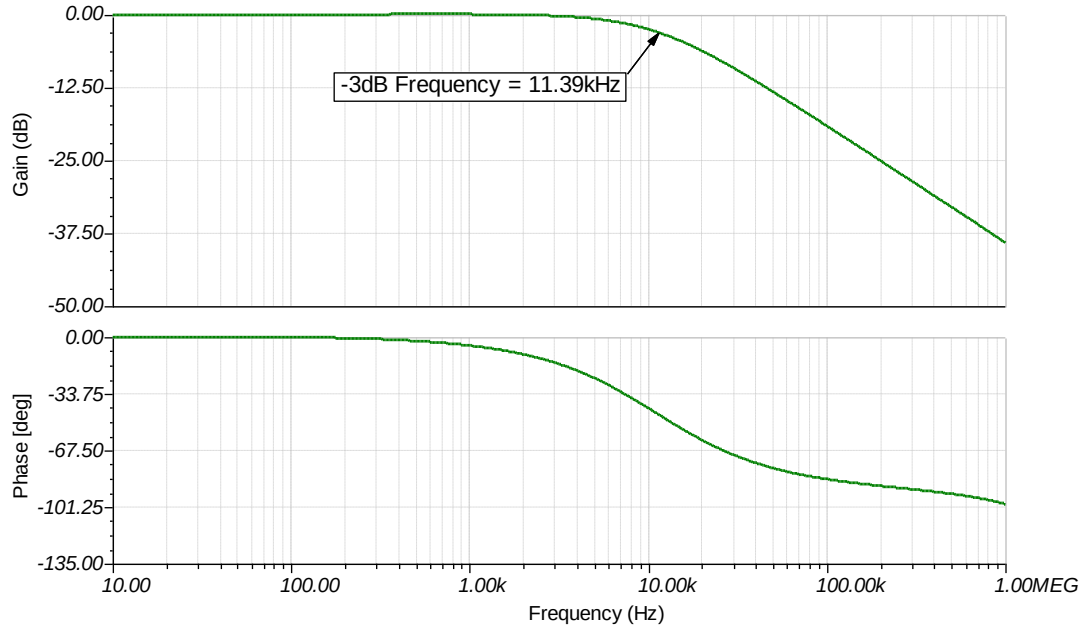
   $200Hz = \frac{1}{2\pi \times R_1 \times 470nF}$, so $R_1 = 1.693\ k\Omega \approx 1.69k\Omega$ (Standard Value)

Design Simulations

Transient Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC508.

See TIPD140, www.ti.com/tool/tipd140.

Design Featured Op Amp

OPA192	
V_{cc}	4.5V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	1mA/Ch
I_b	5pA
UGBW	10MHz
SR	20V/µs
#Channels	1, 2, 4
www.ti.com/product/opa192	

Design Alternate Op Amp

TLV2372	
V_{cc}	2.7V to 16V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	2mV
I_q	750µA/Ch
I_b	1pA
UGBW	3MHz
SR	2.1V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv2372	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

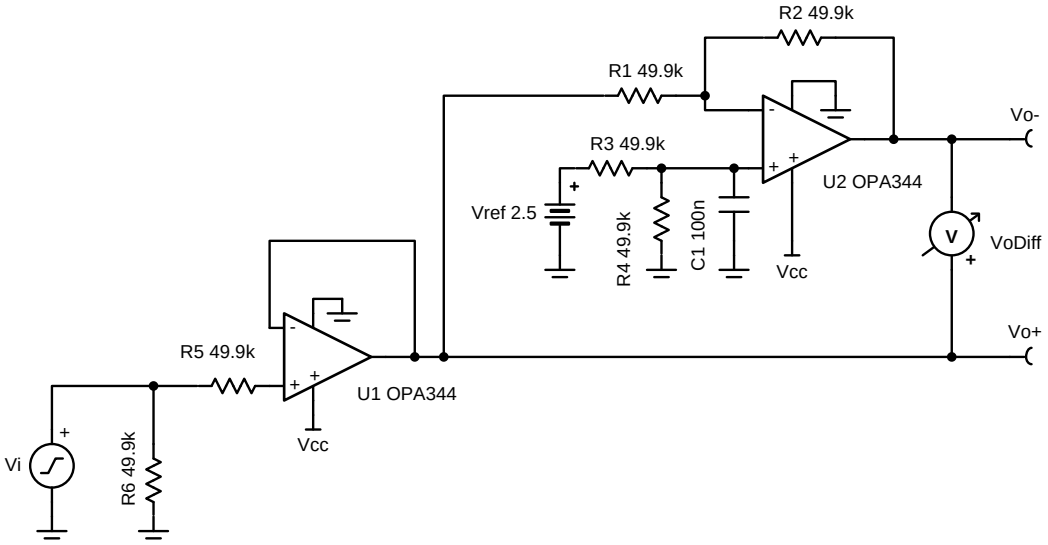
Single-ended input to differential output circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	$V_{oDiffMin}$	$V_{oDiffMax}$	V_{cc}	V_{ee}	V_{ref}
0.1V	2.4V	–2.3V	2.3V	2.7V	0V	2.5V

Design Description

This circuit converts a single ended input of 0.1V to 2.4V into a differential output of ±2.3V on a single 2.7-V supply. The input and output ranges can be scaled as necessary as long as the op amp input common-mode range and output swing limits are met.



Design Notes

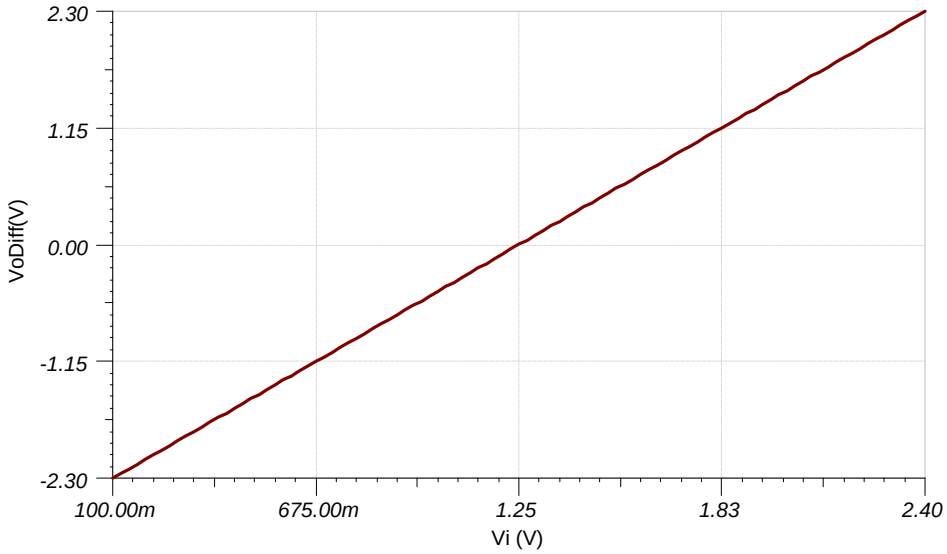
1. Op amps with rail-to-rail input and output will maximize the input and output range of the circuit.
2. Op amps with low V_{os} and offset drift will reduce DC errors.
3. Use low tolerance resistors to minimize gain error.
4. Set output range based on linear output swing (see A_{ol} specification).
5. Keep feedback resistors low or add capacitor in parallel with R_2 for stability.

Design Steps

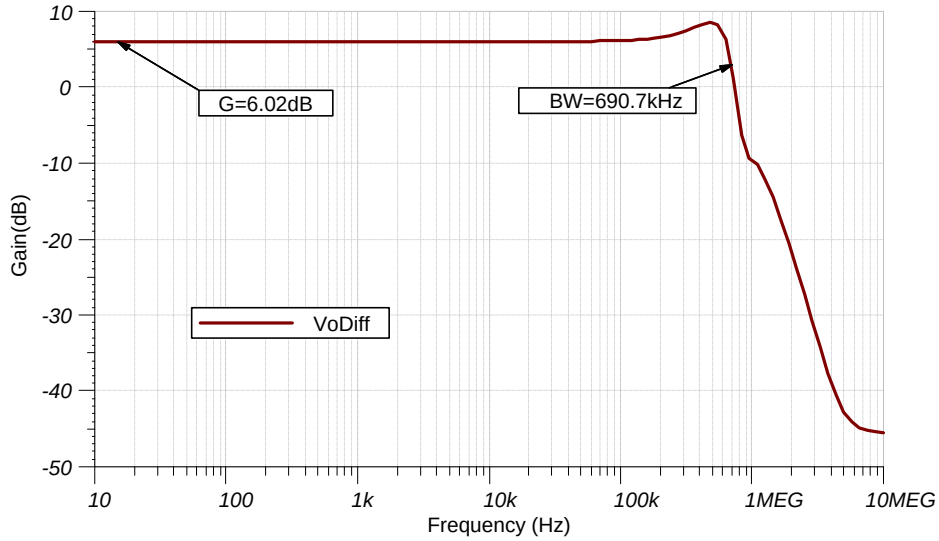
1. Buffer V_i signal to generate V_{o+}.
   $V_{O+} = V_i$
2. Invert and level shift V_{o+} using a difference amplifier to create V_{o-}.
   $V_{o-} = (V_{ref} - V_{o+}) \times (\frac{R_2}{R_1})$
3. Select resistances so that the resistor noise is smaller than the amplifier broadband noise.
   $E_{nv} = 30 \frac{nV}{\sqrt{Hz}}$ (Voltage noise from op amp)
   If $R_1 = R_2 = R_3 = R_4 = 49.9k\Omega$ then
   $E_{nr} = \sqrt{(\sqrt{4 \times kB \times T \times (R_1 || R_2)})^2 + (\sqrt{4 \times kB \times T \times (R_3 || R_4)})^2} = 28.7 \frac{nV}{\sqrt{Hz}} (< E_{nv})$
4. Select resistances that protect the input of the amplifier and prevents floating inputs. To simplify the bill of materials (BOM), select R_5 = R_6.
   $R_5 = R_6 = 49.9k\Omega$

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC510.

See TIPD131, www.ti.com/tool/tipd131.

Design Featured Op Amp

OPA344	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.2mV
I_q	150µA
I_b	0.2pA
UGBW	1MHz
SR	0.8V/µs
#Channels	1, 2, 4
www.ti.com/product/opa344	

Design Alternate Op Amp

OPA335	
V_{ss}	2.7V to 5.5V
V_{inCM}	V_{ee}–0.1V to V_{cc}–1.5V
V_{out}	Rail-to-rail
V_{os}	1µV
I_q	285µA/Ch
I_b	70pA
UGBW	2MHz
SR	1.6V/µs
#Channels	1, 2
www.ti.com/product/opa335	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

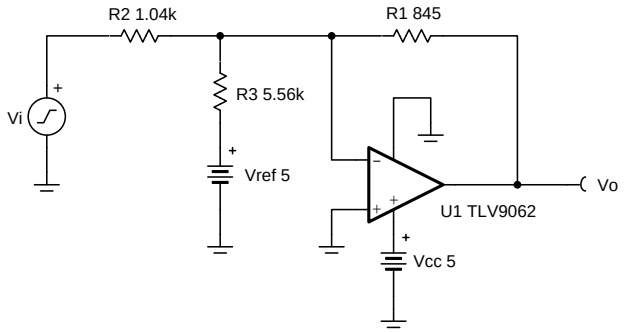
Inverting op amp with inverting positive reference voltage circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–5V	–1V	0.05V	3.3V	5V	0V	5V

Design Description

This design uses an inverting amplifier with an inverting positive reference to translate an input signal of –5V to –1V to an output voltage of 3.3V to 0.05V. This circuit can be used to translate a negative sensor output voltage to a usable ADC input voltage range.



Design Notes

1. Use op amp linear output operating range. Usually specified under A_{OL} test conditions.
2. Common mode range must extend down to or below ground.
3. V_{ref} output must be low impedance.
4. Input impedance of the circuit is equal to R_2.
5. Choose low-value resistors to use in the feedback. It is recommended to use resistor values less than 100kΩ. Using high-value resistors can degrade the phase margin of the amplifier and introduce additional noise in the circuit.
6. The cutoff frequency of the circuit is dependent on the gain bandwidth product (GBP) of the amplifier. Additional filtering can be accomplished by adding a capacitor in parallel to R_1. Adding a capacitor in parallel with R_1 will also improve stability of the circuit if high-value resistors are used.

Design Steps

$$V_o = -V_i \times \left(\frac{R_1}{R_2}\right) - V_{ref} \times \left(\frac{R_1}{R_3}\right)$$

1. Calculate the gain of the input signal.

$$G_{input} = \frac{V_{o\_max} - V_{o\_min}}{V_{i\_max} - V_{i\_min}} = \frac{3.3V - 0.05V}{-1V - (-5\ V)} = 0.8125\frac{V}{V}$$

2. Calculate R_1 and R_2.

Choose $R_1 = 845\Omega$

$$R_2 = \frac{R_1}{G_{input}} = \frac{R_1}{0.8125\frac{V}{V}} = 1.04\ \ k\Omega$$

3. Calculate the gain of the reference voltage required to offset the output.

$$G_{ref} = \frac{R_1}{R_3} \qquad (\quad) \qquad (\quad)$$

$$-V_{i\_min} \times \frac{R_1}{R_2} - V_{ref} \times \frac{R_1}{R_3} = V_{o\_min}$$

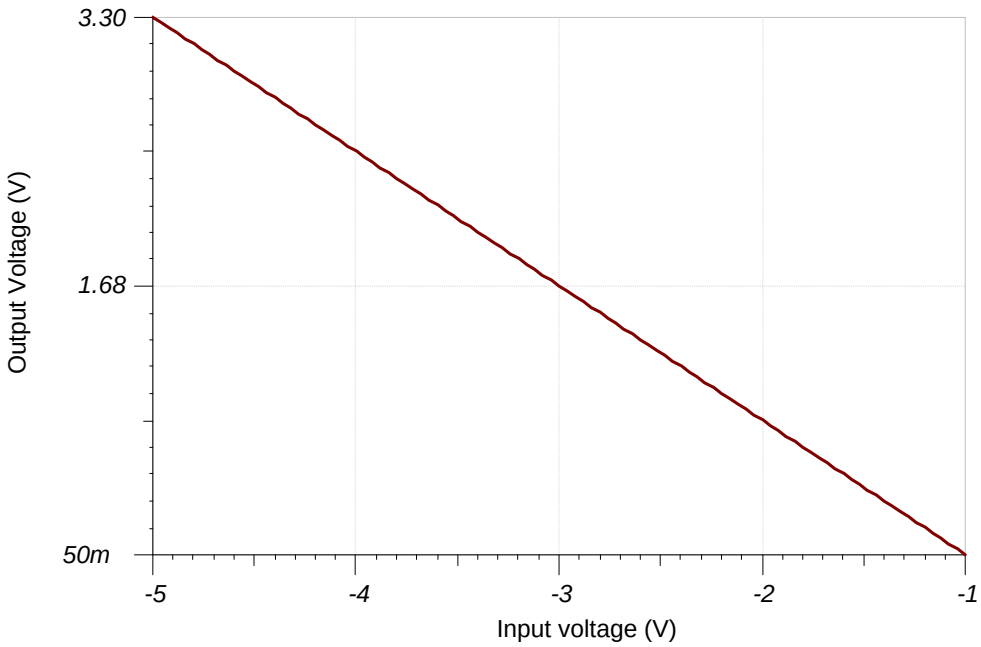
$$\frac{R_1}{R_3} = \frac{V_{o\_min} + V_{i\_min} \times \frac{R_1}{R_2}}{-V_{ref}} = \frac{0.05V + -1\ V \quad \frac{845\Omega}{1.04k\Omega}}{-5} = 0.1525\frac{V}{V}$$

4. Calculate R_3.

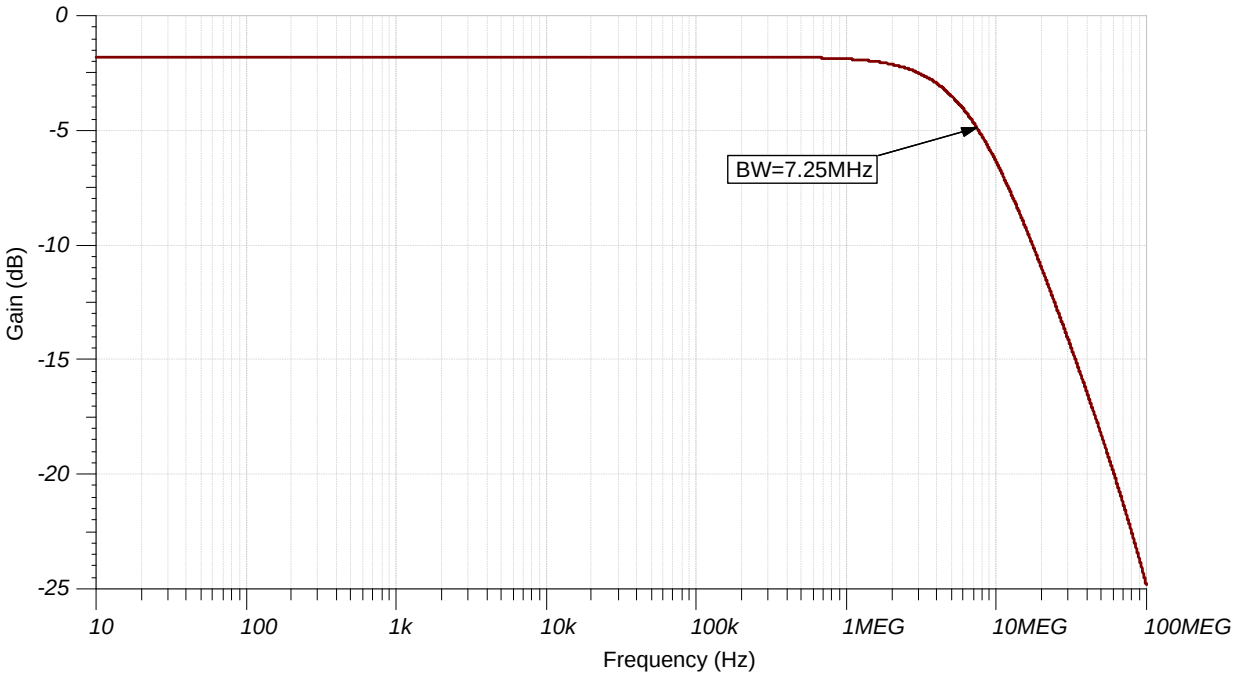
$$R_3 = \frac{R_1}{G_{ref}} = \frac{845\Omega}{0.1525\frac{V}{V}} = 5.54\ \ k\Omega \approx 5.56\ \ k\Omega$$

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC511.

See Designing Gain and Offset in Thirty Seconds .

Design Featured Op Amp

TLV9062	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.3mV
I_q	538µA
I_b	0.5pA
UGBW	10MHz
SR	6.5V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv9062	

Design Alternate Op Amp

OPA197	
V_{ss}	4.5V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	25µV
I_q	1mA
I_b	5pA
UGBW	10MHz
SR	20V/µs
#Channels	1, 2, 4
www.ti.com/product/opa197	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

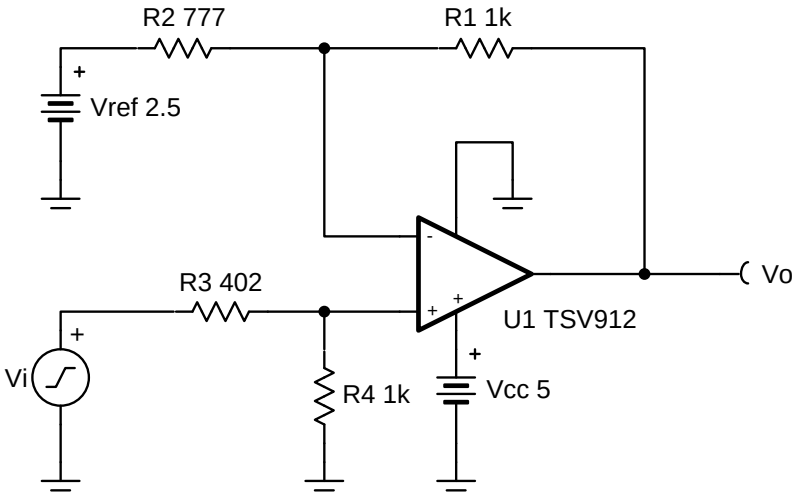
Non-inverting op amp with inverting positive reference voltage circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
2V	5V	0.05V	4.95V	5V	0V	2.5V

Design Description

This design uses a non-inverting amplifier with an inverting positive reference to translate an input signal of 2V to 5V to an output voltage of 0.05V to 4.95V. This circuit can be used to translate a sensor output voltage with a positive slope and offset to a usable ADC input voltage range.



Design Notes

1. Use op amp linear output operating range. Usually specified under A_{OL} test conditions.
2. Check op amp input common mode voltage range. The common mode voltage varies with the input voltage.
3. V_{ref} must be low impedance.
4. Input impedance of the circuit is equal to the sum of R_3 and R_4.
5. Choose low-value resistors to use in the feedback. It is recommended to use resistor values less than 100kΩ. Using high-value resistors can degrade the phase margin of the amplifier and introduce additional noise in the circuit.
6. The cutoff frequency of the circuit is dependent on the gain bandwidth product (GBP) of the amplifier.
7. Adding a capacitor in parallel with R_1 will improve stability of the circuit if high-value resistors are used.

Design Steps

$$V_o = V_i \times \left(\frac{R_4}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right) - V_{ref} \times \left(\frac{R_1}{R_2}\right)$$

1. Calculate the gain of the input to produce the largest output swing.

$$V_{o\_max} - V_{o\_min} = (V_{i\_max} - V_{i\_min})\left(\frac{R_4}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right)$$

$$\frac{V_{o\_max} - V_{o\_min}}{V_{i\_max} - V_{i\_min}} = \frac{R_4}{R_3+R_4}\ \frac{R_1+R_2}{R_2}$$

$$\frac{4.95V - 0.05V}{5V - 2V} = \frac{R_4}{R_3+R_4}\ \frac{R_1+R_2}{R_2}$$

$$1.633\frac{V}{V} = \frac{R_4}{R_3+R_4}\ \frac{R_1+R_2}{R_2}$$

2. Select a value for R_1 and R_4 and insert the values into the previous equation. The other two resistor values must be solved using a system of equations. The proper output swing and offset voltage cannot be calculated if more than two variables are selected.

$$R_1 = R_4 = 1\ \ k\Omega \qquad (\ $$

$$1.633\frac{V}{V} = \frac{1\ k\Omega}{R_3+1\ k\Omega}\ \frac{1\ k\Omega+R_2}{R_2}$$

3. Solve the previous equation for R_3 in terms of R_2.

$$R_3 = \frac{1\ M\Omega+(1\ k\Omega\times R_2)}{1.633\times R_2} - 1\ \ k\Omega$$

4. Select any point along the transfer function within the linear output range of the amplifier to set the proper offset voltage at the output (for example, the minimum input and output voltage).

$$V_{o\_min} = V_{i\_min} \times \left(\frac{R_4}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right) - V_{ref} \times \left(\frac{R_1}{R_2}\right)$$

$$0.05V = 2V \times \frac{1\ k\Omega}{R_3+1\ k\Omega}\ \frac{1\ k\Omega+R_2}{R_2} - V_{ref} \times \frac{1\ k\Omega}{R_2}$$

5. Insert R_3 from step 3 into the equation from step 4 and solve for R_2.

$$0.05V = 2V \times \left(\frac{1\ k\Omega}{\frac{1\ M\Omega+1\ k\Omega\times R_2}{1.633\times R_2} - 1\ k\Omega + 1\ k\Omega}\right)\left(\frac{1\ k\Omega+R_2}{R_2}\right) - V_{ref} \times \left(\frac{1\ k\Omega}{R_2}\right)$$

$$R_2 = 777.2\Omega \approx 777\Omega$$

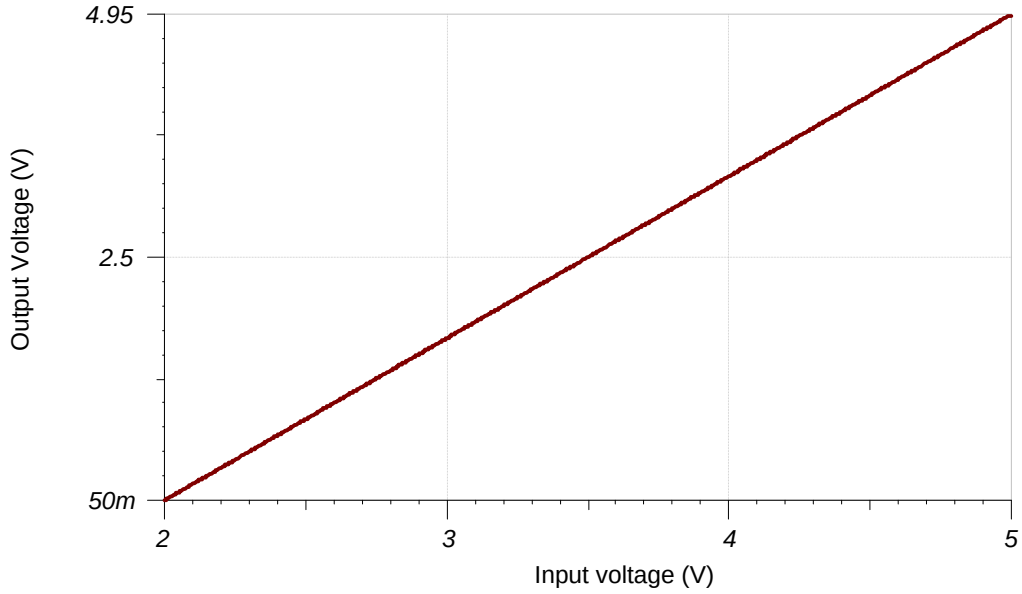
6. Insert R_2 calculation from step 5, and solve for R_3.

$$R_3 = \frac{1\ M\Omega+(1\ k\Omega\times R_2)}{1.633\times R_2} - 1\ \ k\Omega \qquad (\quad)\ (\quad)$$

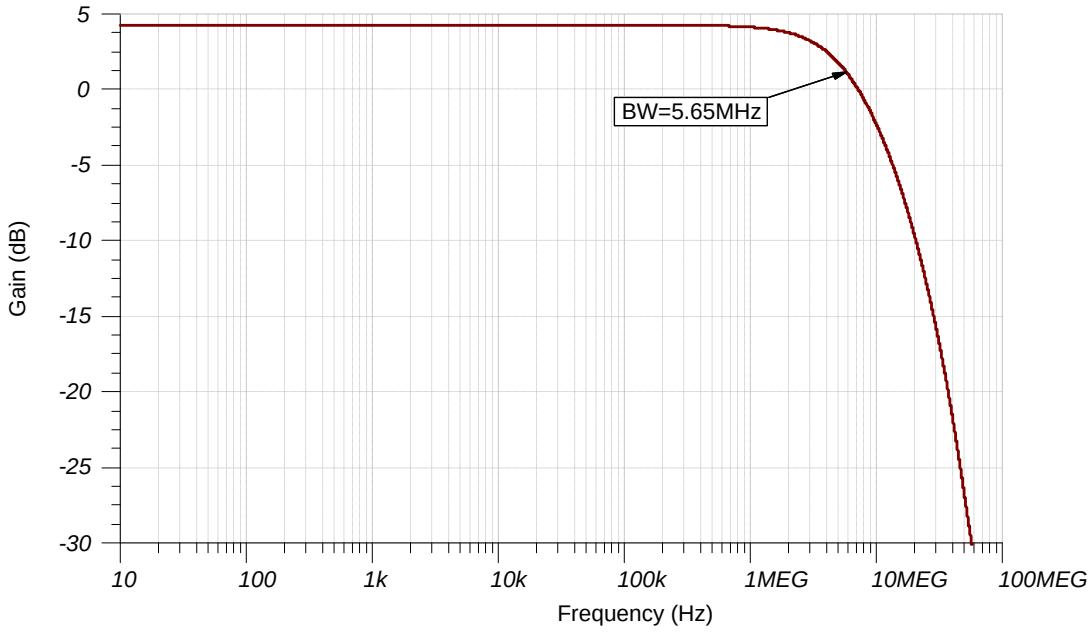
$$R_3 = \frac{1\ M\Omega+1\ k\Omega\times\ 777\Omega}{1.633\times\ 777\Omega} - 1\ \ k\Omega = 400.49\Omega \approx 402\Omega$$

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit SPICE simulation file SBOC512.

See TI Precision Lab Videos on Input and Output Limitations.

Design Featured Op Amp

TSV912	
V_{ss}	2.5V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.3mV
I_q	550µA
I_b	1pA
UGBW	8MHz
SR	4.5V/µs
#Channels	1, 2, 4
www.ti.com/product/tsv912	

Design Alternate Op Amp

OPA191	
V_{ss}	4.5V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	140µA/Ch
I_b	5pA
UGBW	2.5MHz
SR	5.5V/µs
#Channels	1, 2, 4
www.ti.com/product/opa191	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

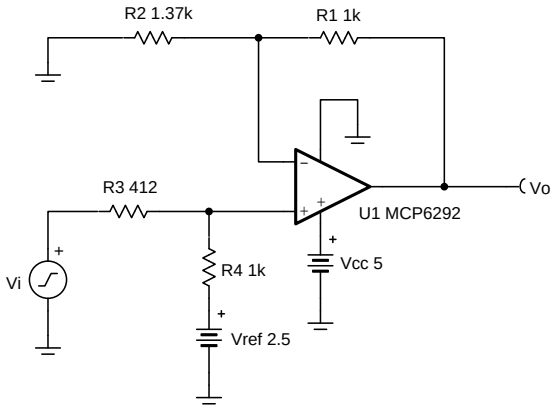
Non-inverting op amp with non-inverting positive reference voltage circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–1V	3V	0.05V	4.95V	5V	0V	2.5V

Design Description

This design uses a non-inverting amplifier with a non-inverting positive reference to translate an input signal of –1V to 3V to an output voltage of 0.05V to 4.95V. This circuit can be used to translate a sensor output voltage with a positive slope and negative offset to a usable ADC input voltage range.



Design Notes

1. Use op amp linear output operating range. Usually specified under A_{OL} test conditions.
2. Check op amp input common mode voltage range.
3. V_{ref} output must be low impedance.
4. Input impedance of the circuit is equal to the sum of R_3 and R_4.
5. Choose low-value resistors to use in the feedback. It is recommended to use resistor values less than 100kΩ. Using high-value resistors can degrade the phase margin of the amplifier and introduce additional noise in the circuit.
6. The cutoff frequency of the circuit is dependent on the gain bandwidth product (GBP) of the amplifier.
7. Adding a capacitor in parallel with R_1 will improve stability of the circuit if high-value resistors are used.

Design Steps

$$V_o = V_i \times \left(\frac{R_4}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right) + V_{ref} \times \left(\frac{R_3}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right)$$

1. Calculate the gain of the input voltage to produce the desired output swing.

$$G_{input} = \left(\frac{R_4}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right) \qquad ($$

$$V_{o\_max} - V_{o\_min} = V_{i\_max} - V_{i\_min} \quad \frac{R_4}{R_3+R_4} \quad \frac{R_1+R_2}{R_2}$$

$$\frac{V_{o\_max} - V_{o\_min}}{V_{i\_max} - V_{i\_min}} = \frac{R_4}{R_3+R_4} \quad \frac{R_1+R_2}{R_2}$$

$$\frac{4.95V - 0.05V}{3V - \ -1V} = \frac{R_4}{R_3+R_4} \quad \frac{R_1+R_2}{R_2}$$

$$1.225V = \frac{R_4}{R_3+R_4} \quad \frac{R_1+R_2}{R_2}$$

2. Select a value for R_1 and R_4 and insert the values into the previous equation. The other two resistor values must be solved using a system of equations. The proper output swing and offset voltage cannot be calculated if more than two variables are selected.

$$R_1 = R_4 = 1 \ \ k\Omega \qquad ($$

$$1.225V = \frac{1\ k\Omega}{R_3+1\ k\Omega} \quad \frac{1\ k\Omega+R_2}{R_2}$$

3. Solve the previous equation for R_3 in terms of R_2.

$$R_3 = \frac{1\ M\Omega + (1\ k\Omega \times R_2)}{1.225 \times R_2} - 1\ \ k\Omega$$

4. Select any point along the transfer function within the linear output range of the amplifier to set the proper offset voltage at the output (for example, the minimum input and output voltage).

$$V_{o\_min} = V_{i\_min} \times \left(\frac{R_4}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right) + V_{ref} \times \left(\frac{R_3}{R_3+R_4}\right)\left(\frac{R_1+R_2}{R_2}\right)$$

$$0.05V = -1 \ \ V \times \frac{1\ k\Omega}{R_3+1\ k\Omega} \quad \frac{1\ k\Omega+R_2}{R_2} + 2.5V \times \frac{R_3}{R_3+1\ k\Omega} \quad \frac{1\ k\Omega+R_2}{R_2}$$

5. Insert R_3 into the equation from step 1 and solve for R_2.

$$0.05V = -1 \ \ V \times \left(\frac{1\ k\Omega}{\frac{1\ M\Omega+1\ k\Omega \times R_2}{1.225 \times R_2} - 1\ k\Omega + 1\ k\Omega}\right)\left(\frac{1\ k\Omega+R_2}{R_2}\right) + 2.5V \times \left(\frac{\frac{1\ M\Omega+1\ k\Omega \times R_2}{1.225 \times R_2} - 1\ k\Omega}{\frac{1\ M\Omega+1\ k\Omega \times R_2}{1.225 \times R_2} - 1\ k\Omega + 1\ k\Omega}\right)\left(\frac{1\ k\Omega+R_2}{R_2}\right)$$

$$R_2 = 1360.5\Omega \ \approx 1370\Omega$$

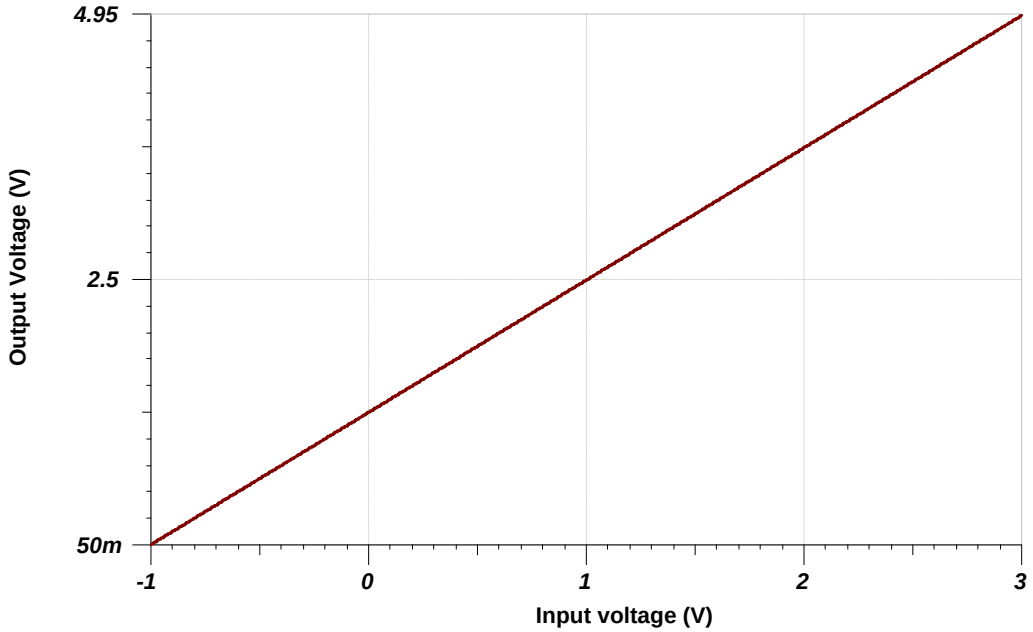
6. Insert R_2 into the equation from step 1 to solve for R_3.

$$R_3 = \frac{1\ M\Omega + 1\ k\Omega \times (1370\Omega)}{1.225 \times (1370\Omega)} - 1\ \ k\Omega$$

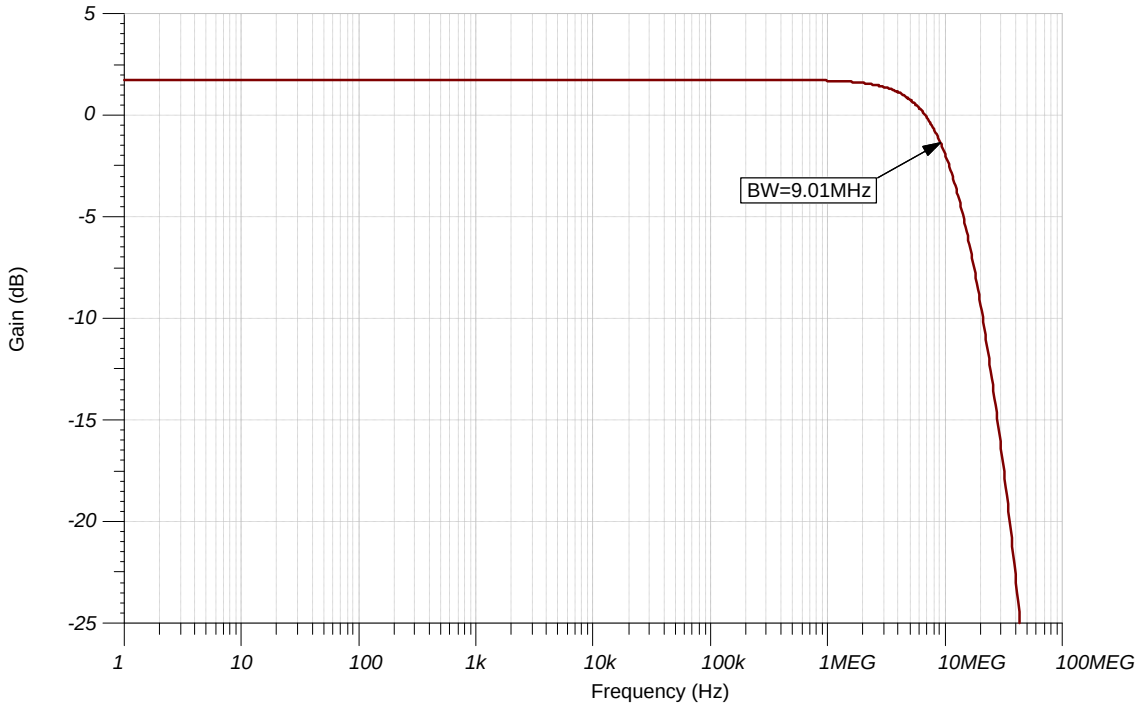
$$R_3 = 412.18\Omega \approx 412\Omega$$

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC513.

See Designing Gain and Offset in Thirty Seconds.

Design Featured Op Amp

MCP6292	
V_{ss}	2.4V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.3mV
I_q	600µA
I_b	1pA
UGBW	10MHz
SR	6.5V/µs
#Channels	1, 2, 4
www.ti.com/product/MCP6292	

Design Alternate Op Amp

OPA388	
V_{ss}	2.5V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.25µV
I_q	1.9mA
I_b	30pA
UGBW	10MHz
SR	5V/µs
#Channels	1, 2, 4
www.ti.com/product/opa388	

Revision History

Revision	Date	Change
A	January 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

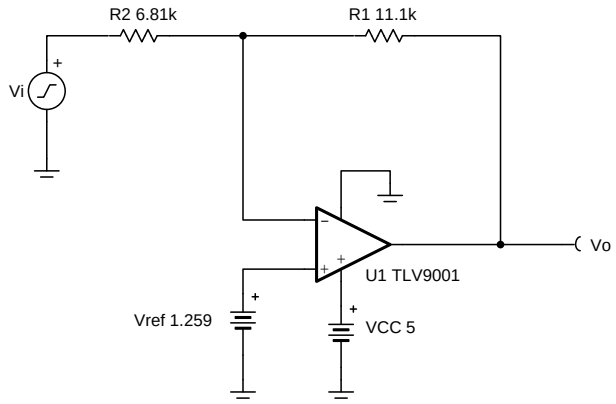
Inverting op amp with non-inverting positive reference voltage circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–1V	2V	0.05V	4.95V	5V	0V	1.259V

Design Description

This design uses an inverting amplifier with a non-inverting positive reference voltage to translate an input signal of –1V to 2V to an output voltage of 0.05V to 4.95V. This circuit can be used to translate a sensor output voltage with a positive slope and negative offset to a usable ADC input voltage range.



Design Notes

1. Use op amp linear output operating range. Usually specified under A_{OL} test conditions.
2. Amplifier common mode voltage is equal to the reference voltage.
3. V_{ref} can be created with a voltage divider.
4. Input impedance of the circuit is equal to R_2.
5. Choose low-value resistors to use in the feedback. It is recommended to use resistor values less than 100kΩ. Using high-value resistors can degrade the phase margin of the amplifier and introduce additional noise in the circuit.
6. The cutoff frequency of the circuit is dependent on the gain bandwidth product (GBP) of the amplifier. Additional filtering can be accomplished by adding a capacitor in parallel to R_1. Adding a capacitor in parallel with R_1 will also improve stability of the circuit, if high-value resistors are used.

Design Steps

$$V_o = -Vi \times \left(\frac{R_1}{R_2}\right) + V_{ref} \times \left(1 + \frac{R_1}{R_2}\right)$$

1. Calculate the gain of the input signal.

$$G_{input} = -\frac{R_1}{R_2} \qquad (\qquad)(\qquad)$$

$$V_{o\_max} - V_{o\_min} = V_{i\_max} - V_{i\_min} \quad -\frac{R_1}{R_2}$$

$$-\frac{R_1}{R_2} = -\frac{V_{o\_max} - V_{o\_min}}{V_{i\_max} - V_{i\_min}} = -\frac{4.95V - 0.05V}{2V - \ -1\ V} = -1.633\frac{V}{V}$$

2. Select R_2 and calculate R_1.

$$R_2 = 6.81 \ \ k\Omega$$

$$R_1 = G_{input} \times R_2 = 1.633\frac{V}{V} \times 6.81 \ \ k\Omega = 11.123k\Omega \approx 11.1 \ \ k\Omega \ \ \text{(Standard Value)}$$

3. Calculate the reference voltage.

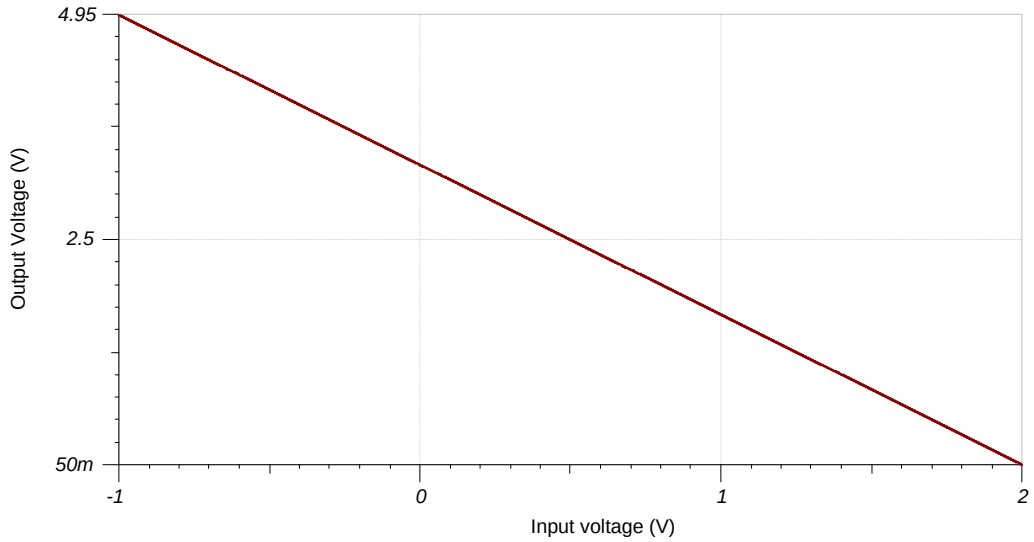
$$V_{o\_min} = -V_{i\_max} \times \left(\frac{R_1}{R_2}\right) + V_{ref} \times \left(1 + \frac{R_1}{R_2}\right)$$

$$0.05V = -2V \times \frac{11.11\ k\Omega}{6.81\ k\Omega} + V_{ref} \times \left(1 + \frac{11.11\ k\Omega}{6.81\ k\Omega}\right)$$

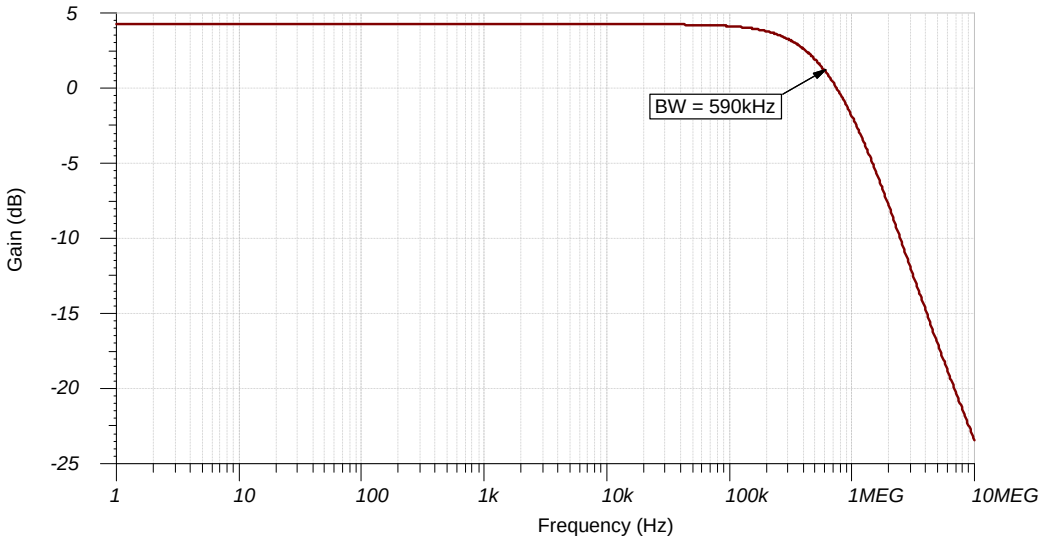
$$V_{ref} = \frac{V_{o\_min} + V_{i\_max} \times \frac{R_1}{R_2}}{1 + \frac{R_1}{R_2}} \quad \frac{0.05V + 2V \times \frac{11.11\ k\Omega}{6.81\ k\Omega}}{1 + \frac{11.11\ k\Omega}{6.81\ k\Omega}} = 1.259V$$

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC514.

See the Designing gain and offset in thirty seconds application report.

Design Featured Op Amp

TLV9001	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.4mV
I_q	60µA
I_b	5pA
UGBW	1MHz
SR	2V/µs
#Channels	1, 2, 4
www.ti.com/product/tlv9002	

Design Alternate Op Amp

OPA376	
V_{ss}	2.2V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	760µA
I_b	0.2pA
UGBW	5.5MHz
SR	2V/µs
#Channels	1, 2, 4
www.ti.com/product/opa376	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

Single-supply diff-in to diff-out AC amplifier circuit

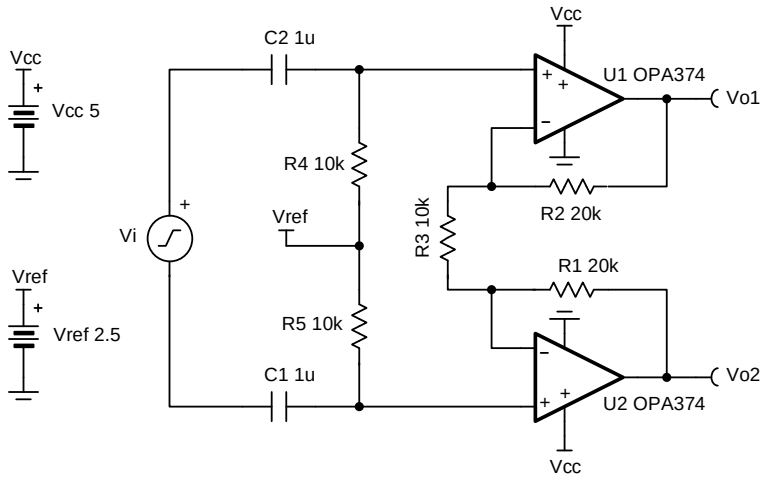
Design Goals

Diff. Input V_i		Diff. Output ($V_{o1} - V_{o2}$)		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–500mV	+500mV	–2.5V	+2.5V	+5	0V	+2.5V

Lower Cutoff Freq.	Upper Cutoff Freq.
16Hz	> 1MHz

Design Description

This circuit uses 2 op amps to build a discrete, single-supply diff-in diff-out amplifier. The circuit converts a differential signal to a differential output signal.



Design Notes

1. Ensure that R_1 and R_2 are well matched with high accuracy resistors to maintain high DC common-mode rejection performance.
2. Increase R_4 and R_5 to match the necessary input impedance at the expense of thermal noise performance.
3. Bias for single-supply operation can also be created by a voltage divider from V_{cc} to ground.
4. V_{ref} sets the output voltage of the instrumentation amplifier bias at mid-supply to allow the output to swing to both supply rails.
5. Choose C_1 and C_2 to select the lower cutoff frequency.
6. Linear operation is contingent upon the input common-mode and the output swing ranges of the discrete op amps used. The linear output swing ranges are specified under the Aol test conditions in the op amps data sheets

Design Steps

1. The transfer function of the circuit is shown below.

   $V_{oDiff} = V_i \times G + V_{ref}$

   where V_i = the differential input voltage

   V_{ref} = the reference voltage provided to the amplifier

   $G = 1 + 2 \times (\frac{R_1}{R_3})$

2. Choose resistors R_1 = R_2 to maintain common-mode rejection performance.

   Choose $R_1 = R_2 = 20\ k\Omega$ (Standard value)

3. Choose resistors R_4 and R_5 to meet the desired input impedance.

   Choose $R_4 = R_5 = 10\ k\Omega$ (Standard value)

4. Calculate R_3 to set the differential gain.

   $Gain = 1 + (\frac{2 \times R_1}{R_3}) = 5\ \frac{V}{V}$

   $R_1 = R_2 = 20\ k\ \Omega$

   $G = 1 + \frac{2 \times 20\ k\Omega}{R_3} = 5\ \frac{V}{V} \rightarrow 5\ \frac{V}{V} - 1 = \frac{40\ k\Omega}{R_3} = 4 \rightarrow R_3 = \frac{40\ k\Omega}{4} = 10\ k\Omega$ (Standard value)

5. Set the reference voltage V_{ref} at mid-supply.

   $V_{ref} = \frac{V_{cc}}{2} = \frac{5\ V}{2} \rightarrow V_{ref} = 2\ .\ 5V$

6. Calculate C_1 and C_2 to set the lower cutoff frequency.

   $f_c = \frac{1}{2 \times \pi \times R_4 \times C_1} = 16\ Hz$

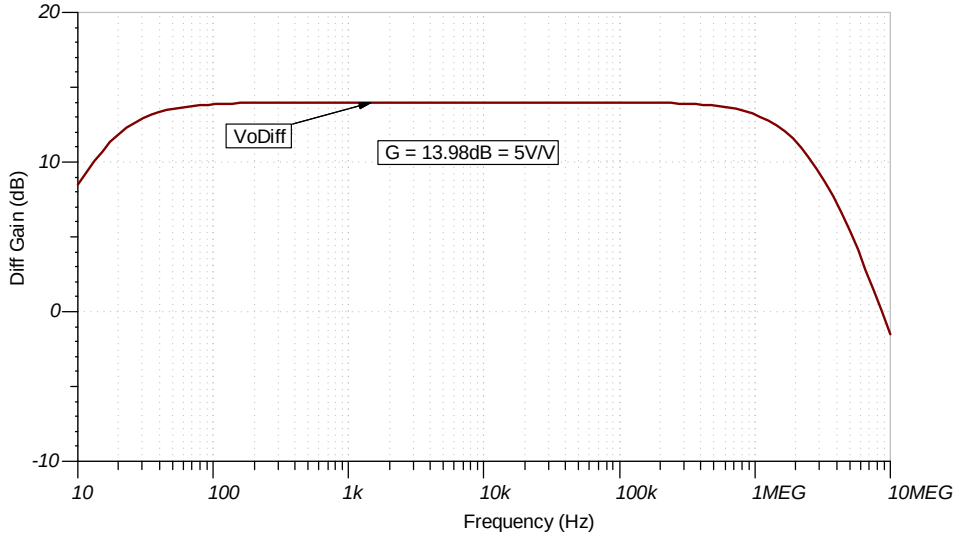
   $R_4 = R_5 = 10\ k\Omega$

   $f_c = \frac{1}{2 \times \pi \times 10\ k\Omega \times C_1} = 16\ Hz \rightarrow C_1 = \frac{1}{2 \times \pi \times 10\ k\Omega \times 16\ Hz} = 0\ .\ 99\mu F \rightarrow C_1 = C_2 = 1\mu F$ (Standard value)

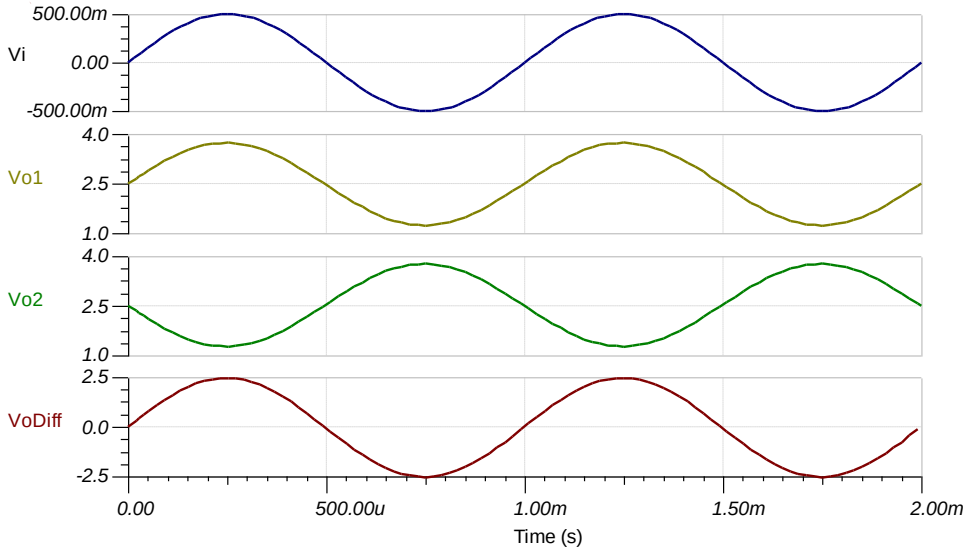
Design Simulations

AC Simulation Results

In the following figure, notice the lower –3-dB cutoff frequency is approximately 16Hz and the upper cutoff frequency is > 1MHz as required for this design.



Transient Simulation Results



References

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAU5.
3. TI Precision Labs

Design Featured Op Amp

OPA374	
V_{ss}	2.3V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1mV
I_q	585µA/Ch
I_b	0.5pA
UGBW	6.5MHz
SR	5V/µs
#Channels	1,2,4
www.ti.com/product/opa374	

Design Alternate Op Amp

TLV9061	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.3mV
I_q	0.538mA
I_b	0.5pA
UGBW	10MHz
SR	6.5V/µs
#Channels	1,2,4
www.ti.com/product/tlv9061	

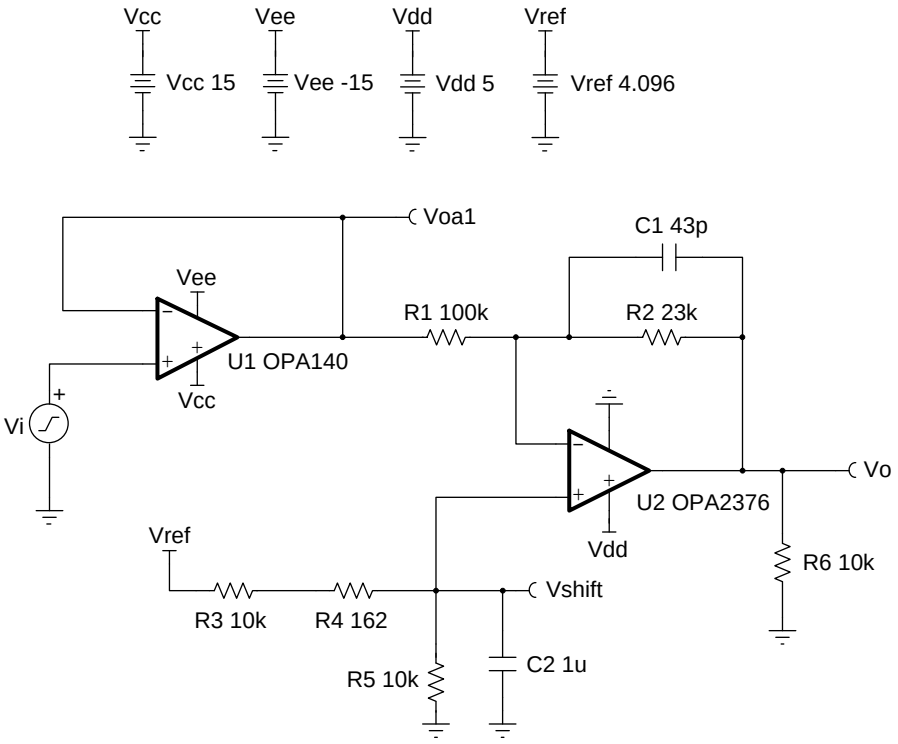
Inverting dual-supply to single-supply amplifier circuit

Design Goals

Input		Output		Supply			
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{dd}	V_{ref}
–10V	+10V	+0.2V	+4.8V	+15V	–15V	+5V	+4.096V

Design Description

This inverting dual-supply to single-supply amplifier translates a ±10-V signal to a 0-V to 5-V signal for use with an ADC. Levels can easily be adjusted using the given equations. The buffer can be replaced with other ±15-V configurations to accommodate the desired input signal, as long as the output of the first stage is low impedance.



Design Notes

1. Observe common-mode limitations of the input buffer.
2. A high-impedance source will alter the gain characteristics of U_2 if buffer amplifier U1 is not used.
3. R_6 provides a path to ground for the output of U_1 if the ±15-V supplies come up before the 5-V supply. This limits the voltage at the inverting pin of U_2 through the voltage divider created by R_1, R_2, and R_6 and prevents damage to U_2 as well as to any converter that may be connected to its output. To best protect the devices a transient voltage suppressor (TVS) should be used at the power pins of U_2.
4. A capacitor across R_5 will help filter V_{ref} and provide a cleaner V_{shift}.

Design Steps

The transfer function for this circuit follows:

$$V_o = -\frac{R_2}{R_1} \times V_i + \left(1 + \frac{R_2}{R_1}\right) \times V_{shift}$$

1. Set the gain of the amplifier.

   $$\frac{\Delta V_o}{\Delta V_i} = \frac{V_{oMax} - V_{oMin}}{V_{iMax} - V_{iMin}} = \frac{4.8\ V - 0.2\ V}{10\ V - (-10\ V)} = 0.23$$

   $$\frac{\Delta V_o}{\Delta V_i} = \frac{R_2}{R_1}$$

   $$R_2 = 0.23 \times R_1$$

   Choose $R_1 = 100k\Omega$ (standard value)

   $R_2 = 23k\Omega$ (for standard values use $22k\Omega$ and $1k\Omega$ in series)

2. Set V_{shift} to translate the signal to single supply.

   At midscale, $V_{in} = 0V$

   Then $V_o = \left(1 + \frac{R_2}{R_1}\right) \times V_{shift}$

   $$V_{shift} = \frac{V_o}{\left(1 + \frac{R_2}{R_1}\right)} = \frac{2.5V}{1.23} = 2.033V$$

3. Select resistors for reference voltage divider to achieve V_{shift}.

   $V_{ref} = 4.096V$

   $$V_{shift} = V_{ref} \times \frac{R_5}{(R_3 + R_4) + R_5}$$

   $$\frac{V_{shift}}{V_{ref}} = \frac{2.033V}{4.096V} = \frac{R_5}{(R_3 + R_4) + R_5}$$

   $$R_3 + R_4 = 1.0161 \times R_5$$

   Select a standard value for R_5

   $R_5 = 10k\Omega$

   $R_3 + R_4 = 10.161k\Omega$

   $R_3 = 10k\Omega$

   $R_4 = 162\Omega$ (standard 1% value)

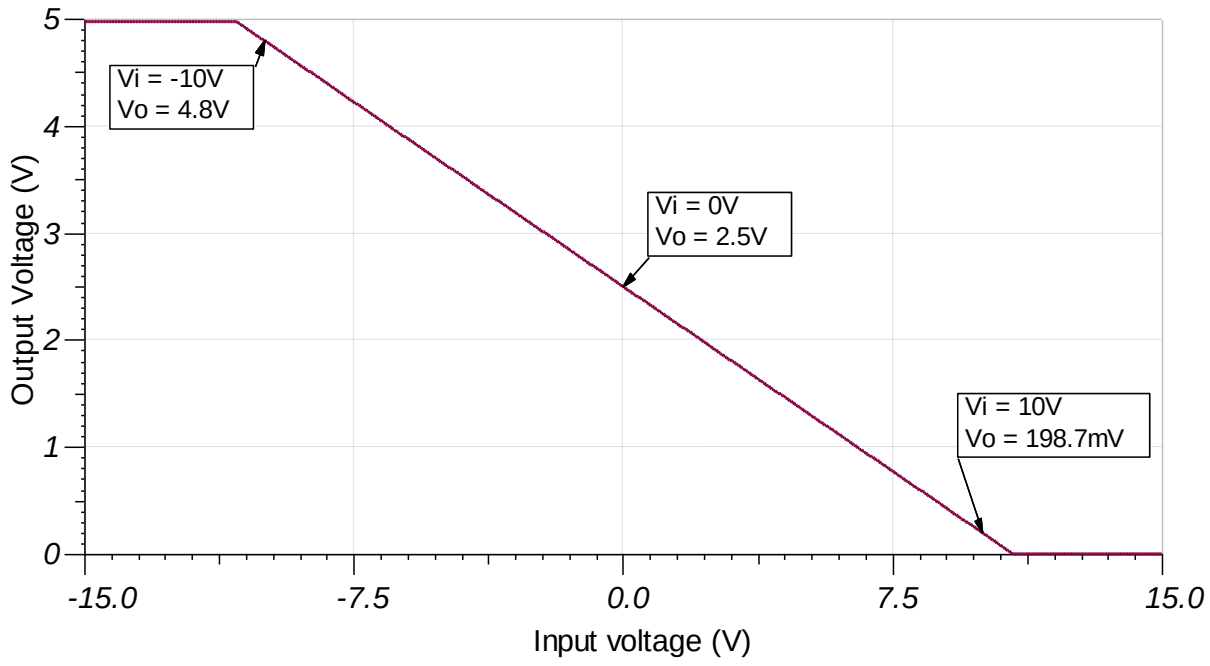
4. Large feedback resistors can interact with the input capacitance and cause instability. Choose C_1 to add a pole to the transfer function to counteract this. The pole must be lower in frequency than the effective bandwidth of the op amp.

   $C_1 = 43pF$

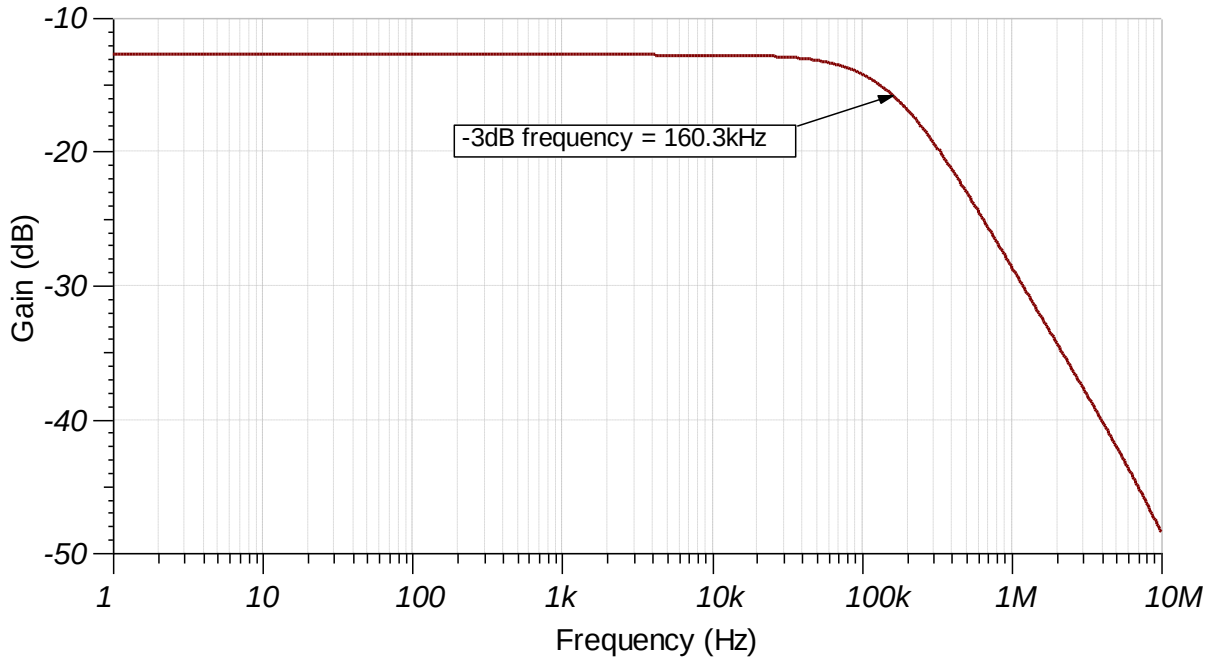
   $$f_p = \frac{1}{2\pi \times R_2 \times C_1} = 160.3kHz$$

Design Simulations

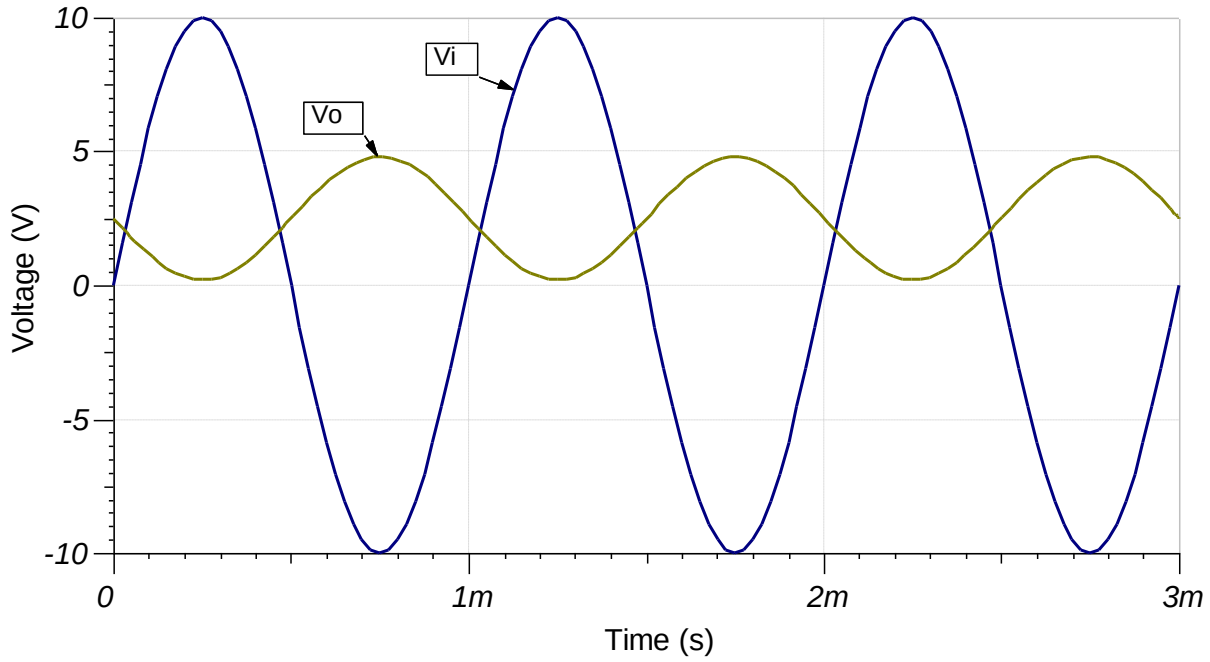
DC Simulation Results



AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See TINA-TI™ circuit simulation file, SBOMAT9.

See TIPD148, http://www.ti.com/tool/TIPD148.

Design Featured Op Amp

OPA376	
V_{ss}	2.2V to 5.5V
V_{inCM}	Vee to Vcc-1.3V
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	760µA/Ch
I_b	0.2pA
UGBW	5.5MHz
SR	2V/µs
#Channels	1,2,4
http://www.ti.com/product/opa376	

Design Featured Op Amp

OPA140	
V_{ss}	4.5V to 36V
V_{inCM}	Vee-0.1V to Vcc-3.5V
V_{out}	Rail-to-rail
V_{os}	30µV
I_q	1.8mA/Ch
I_b	±0.5pA
UGBW	11MHz
SR	20V/µs
#Channels	1,2,4
http://www.ti.com/product/opa375	

Dual-supply, discrete, programmable gain amplifier circuit

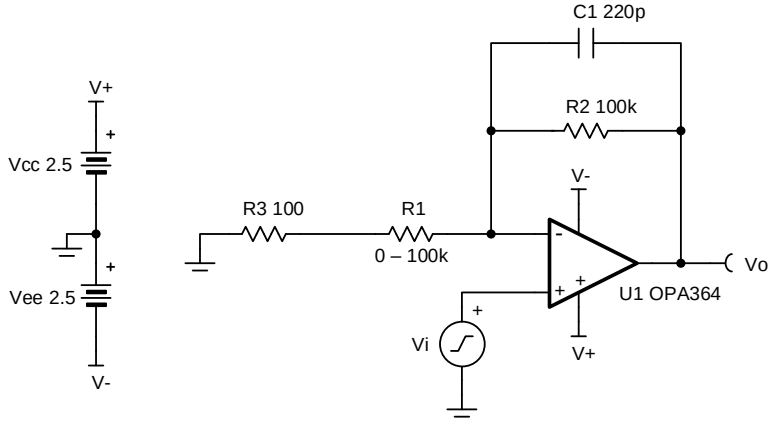
Design Goals

Input		Output		Supply	
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}
–1.25V	+1.25V	–2.4V	+2.4V	+2.5V	–2.5V

Gain	Cutoff Frequency
6dB (2V/V) to 60dB (1000 V/V)	7kHz

Design Description

This circuit provides programmable, non-inverting gains ranging from 6dB (2V/V) to 60dB (1000V/V) using a variable input resistance. The design maintains the same cutoff frequency over the gain range.



Design Notes

1. Choose a digital potentiometer, such as TPL0102 for R_1 to design a low-cost digital programmable gain amplifier.
2. R_3 sets the maximum gain when R_1 approaches 0Ω.
3. A feedback capacitor limits the bandwidth and prevent stability issues.
4. Stability should be evaluated across the selected gain range. The minimum gain setting will likely be most sensitive to stability issues.
5. Some digital potentiometers can vary in absolute value by as much as +/-20% so gain calibration may be necessary.

Design Steps

1. Choose R_2 and R_3, to set the maximum gain when R_1 approaches 0:

$$G_{max} = 1 + \frac{R_2}{R_3}$$

$$G_{max} - 1 = \frac{R_2}{R_3} \rightarrow R_2 = (G_{max} - 1) \times R_3$$

$$\text{Set} \quad R_3 = 100\ \Omega$$

$$R_2 = (1000\ \frac{V}{V} - 1) \times 100 = 99\ k\Omega \rightarrow R_2 = 100\ k\Omega \quad (\text{Standard} \quad \text{value})$$

2. Choose the potentiometer maximum value to set the minimum gain:

$$G_{min} = 1 + \frac{R_2}{R_{1,max} + R_3}$$

$$G_{min} - 1 = \frac{R_2}{R_{1,max} + R_3}$$

$$R_{1,max} + R_3 = \frac{R_2}{G_{min} - 1}$$

$$R_{1,max} = \frac{R_2}{G_{min} - 1} - R_3 = \frac{100k\Omega}{2 - 1} - 100\Omega = 99.9k\Omega \rightarrow R_{1,max} = 100k\Omega \quad (\text{Standard} \quad \text{value})$$

$$R_{1,min} = 0\Omega \quad (\text{Wiper} \quad \text{resistance,} \quad \text{typically} \quad 25\Omega, \quad \text{will} \quad \text{introduce} \quad \text{some} \quad \text{error})$$

3. Choose the bandwidth with a feedback capacitor:

$$f_c = \frac{GBW}{G_{max}} = \frac{7MHz}{1000\frac{V}{V}} = 7kHz$$

$$f_c = 7kHz \rightarrow C_1 = \frac{1}{2\pi \times R_2 \times f_c} = 227pF \quad \rightarrow C_1 = 220pF \quad (\text{Standard} \quad \text{Value})$$

4. Check for stability at minimum gain (2V/V), which is when R_1=100kΩ. To satisfy the requirement f_c (circuit bandwidth) must be less than f_{zero} (zero created by the resistive feedback network and the differential and common-mode input capacitances).

$$f_c = \frac{1}{2\pi \times C_1 \times R_2} = 7\ kHz$$

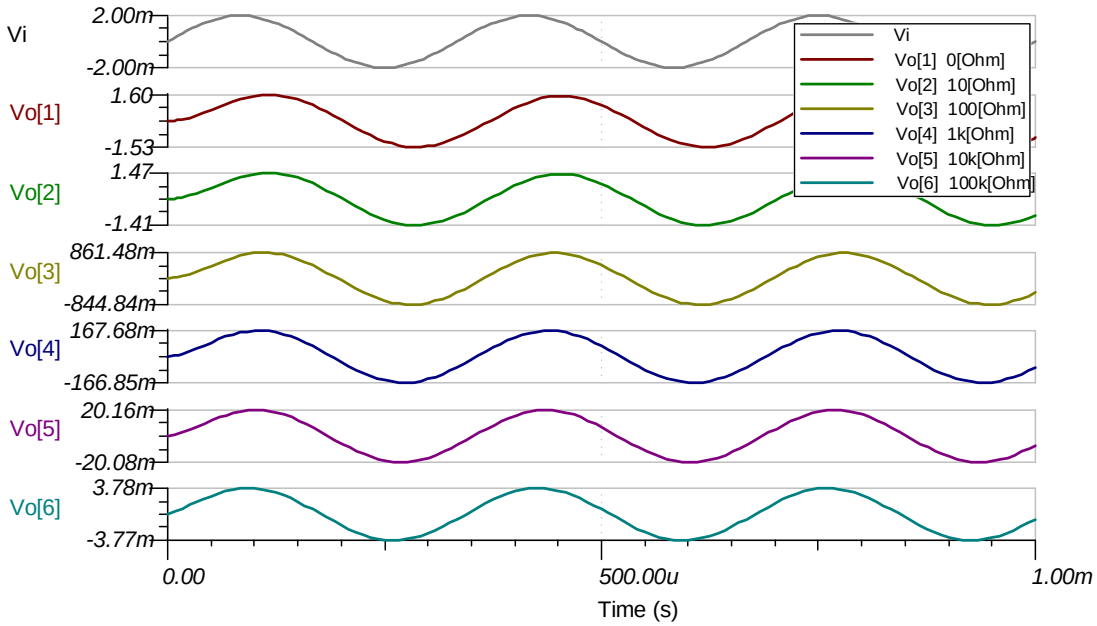
$$f_{zero} = \frac{1}{2\pi \times (C_{cm} + C_{diff}) \times (R_2 \parallel R_1)} = \frac{1}{2 \times \pi \times \left(3\ pF + 2\ pF\right) \times \left(\frac{100\ k\Omega \times 100\ k\Omega}{100\ k\Omega + 100\ k\Omega}\right)}$$

$$f_{zero} = 637\ kHz$$

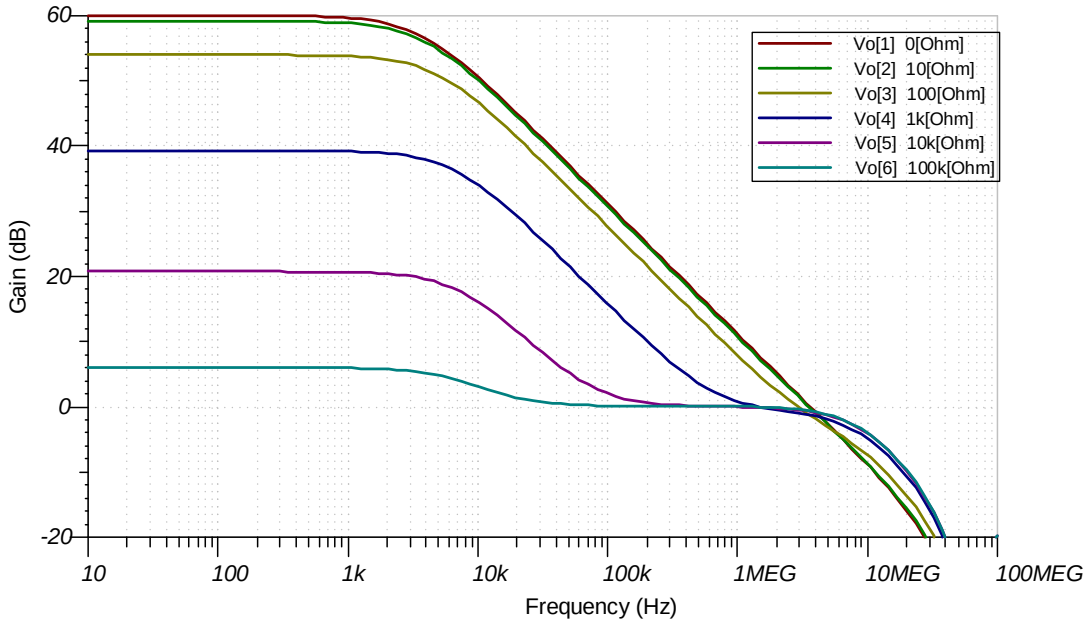
$$7\ kHz < 637\ kHz \rightarrow f_c < f_{zero}$$

Design Simulations

Transient Simulation Results



AC Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOC521
3. TI Precision Designs TIPD204
4. TI Precision Labs

Design Featured Op Amp

OPA364	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1mV
I_q	1.1mA
I_b	1pA
UGBW	7MHz
SR	5V/µs
#Channels	1, 2, 4
www.ti.com/product/opa364	

Design Alternate Op Amp

OPA376	
V_{ss}	2.2V to 5.5V
V_{inCM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	5µV
I_q	760µA
I_b	0.2pA
UGBW	5.5MHz
SR	2V/µs
#Channels	1, 2, 4
www.ti.com/product/opa376	

AC coupled instrumentation amplifier circuit

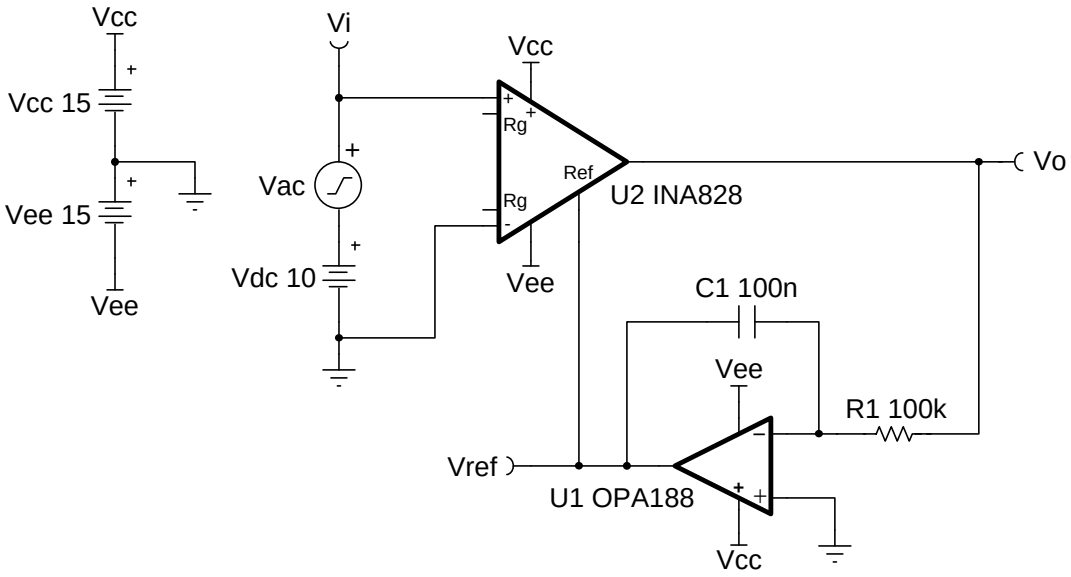
Design Goals

Input		Output		Supply	
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}
–13V	13V	–14.85V	14.85	15	–15

Lower Cutoff Frequency (f_L)	Gain	Input
16Hz	1	±2VAC; +10VDC

Design Description

This circuit produces an AC-coupled output from a DC-coupled input to an instrumentation amplifier. The output is fed back through an integrator, and the output of the integrator is used to modulate the reference voltage of the amplifier. This creates a high-pass filter and effectively cancels the output offset. This circuit avoids the need for large capacitors and resistors on the input, which can significantly degrade CMRR due to component mismatch.



Design Notes

1. The DC correction from output to reference is unity-gain. U_1 can only correct for a signal within its input/output limitations, thus the magnitude of DC voltage that can be corrected for will degrade with increasing instrumentation amplifier gain. See the table in Design Steps for more information.
2. Large values of R_1 and C_1 will lower the cutoff frequency, but increase startup transient response time. Startup behavior can be observed in the Transient Simulation Results.
3. When AC-coupling this way, the total input voltage must remain within the common-mode input range of the instrumentation amplifier.

Design Steps

1. Set the lower cutoff frequency for circuit (integrator cutoff frequency). The upper cutoff frequency will be dictated by the gain and instrumentation amplifier bandwidth.

   $$f_L = \frac{1}{2\pi \times R_1 \times C_1} = 16 \text{ Hz}$$

2. Choose a standard value for R_1 and C_1.

   $$C_1 = 100 \text{ nF}$$

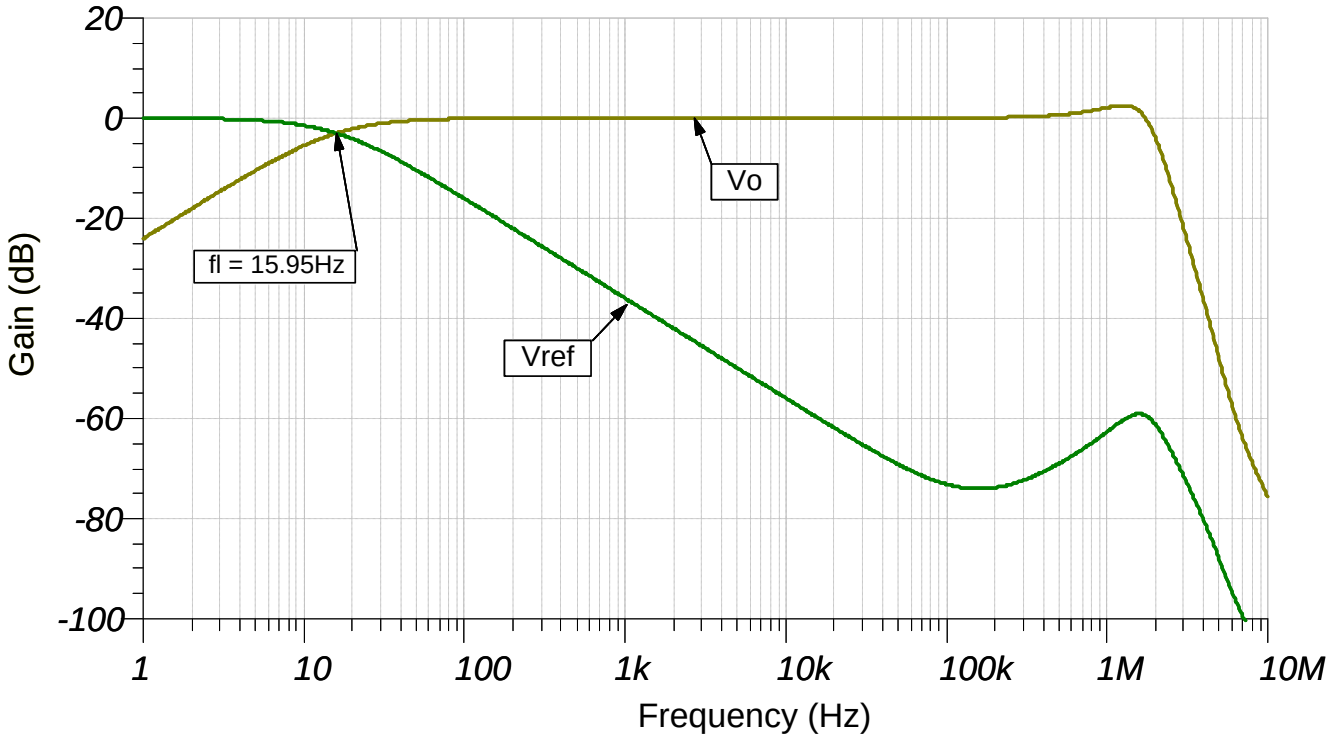
   $$R_1 = \frac{1}{2\pi \times 100 \text{ nF} \times 16 \text{ Hz}} = 99.47 \text{ k}\Omega \approx 100 \text{ k}\Omega \text{ (standard value)}$$

3. The DC rejection capabilities of the circuit will degrade with gain. The following table provides a good estimate of the DC correction range for higher gains.

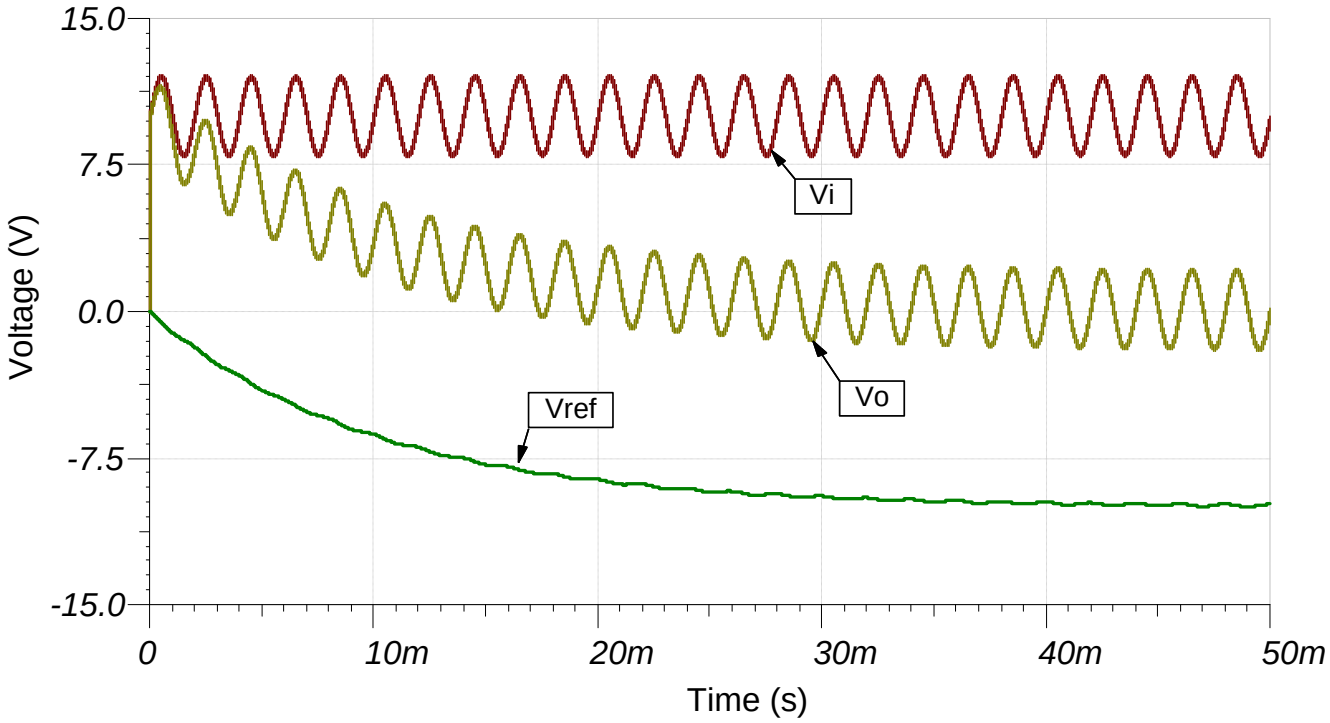
Gain	DC Correction Range
1V/V	±10V
10V/V	±1V
100V/V	±0.1V
1000V/V	±0.01V

Design Simulations

AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See TINA-TI™ circuit simulation file, SBOMAU0.

See TIPD191, http://www.ti.com/tool/tipd191.

Design Featured Instrumentation Amplifier

INA828	
V_{ss}	4.5V to 36V
V_{inCM}	V_{ee}+2V to V_{cc}–2V
V_{out}	V_{ee}+150mV to V_{cc}–150mV
V_{os}	20µV
I_q	600µA
I_b	150pA
UGBW	2MHz
SR	1.2V/µs
#Channels	1
www.ti.com/product/INA828	

Design Featured Op Amp

OPA188	
V_{ss}	8V to 36V
V_{inCM}	V_{ee} to V_{cc}–1.5V
V_{out}	Rail-to-rail
V_{os}	6µV
I_q	450µA
I_b	±160pA
UGBW	2MHz
SR	0.8V/us
#Channels	1,2,4
www.ti.com/product/OPA188	

Design Alternate Op Amp

TLV171	
V_{ss}	2.7V to 36V
V_{inCM}	V_{ee}–0.1V to V_{cc}–2V
V_{out}	Rail-to-rail
V_{os}	750µV
I_q	525µA
I_b	±10pA
UGBW	3MHz
SR	1.5V/us
#Channels	1,2,4
www.ti.com/product/OPA188	

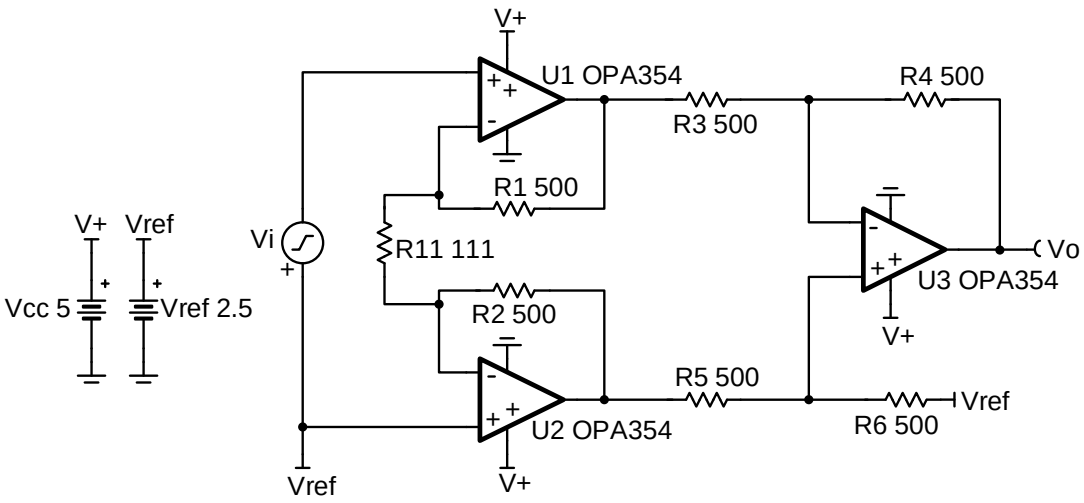
Discrete wide bandwidth INA circuit

Design Goals

Input		Output		Bandwidth	Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	BW	V_{cc}	V_{ee}	V_{ref}
–0.24V	+0.24V	+0.1V	+4.9V	10MHz	+2.5V	0V	2.5V

Design Description

This design uses 3 op–amps to build a discrete wide bandwidth instrumentation amplifier. The circuit converts a differential, high frequency signal to a single–ended output.



Design Notes

1. Reduce the capacitance on the output of each op amp to avoid stability issues.
2. Use low gain configurations to maximize the bandwidth of the circuit.
3. Use precision resistors to achieve high DC CMRR performance.
4. Use small resistors in op–amp feedback to maintain stability.
5. Set the reference voltage, V_{ref}, at mid–supply to allow the output to swing to both supply rails.
6. Phase margin of 45° or greater is required for stable operation.
7. R_{11} sets the gain of the instrumentation amplifier.
8. Linear operation depends upon the input common–mode and the output swing ranges of the discrete op amps used. The linear output swing ranges are specified under the A_{ol} test conditions in the op amps datasheets.
9. V_{ref} also sets the common-mode voltage of the input, V_i, to ensure linear operation.

Design Steps

1. The transfer function of the circuit is given below.

$$V_o = V_i \times \left(1 + \frac{2 \times R_1}{R_{11}}\right) \times \left(\frac{R_6}{R_5}\right) + V_{ref}$$

where V_i = the differential input voltage

V_{ref} = the reference voltage provided to the amplifier

$$\text{Gain} = \left(1 + \frac{2 \times R_1}{R_{11}}\right) \times \frac{R_6}{R_5}$$

2. To maximize the usable bandwidth of design, set the gain of the diff amp stage to 1V/V. Use smaller value resistors to minimize noise.

Choose $R_3 = R_4 = R_5 = R_6 = 500\Omega$ (Standard value)

3. Choose values for resistors R_1 and R_2. Keep these values low to minimize noise.

$R_1 = R_2 = 500\,\Omega$ (Standard value)

4. Calculate resistor R_{11} to set the gain of the circuit to 10V/V

$$G = \left(1 + \frac{2 \times R_1}{R_{11}}\right) = 10\frac{V}{V} \rightarrow 1 + \frac{2 \times 500\,\Omega}{R_{11}} = 10\frac{V}{V} \rightarrow \frac{2 \times 500\,\Omega}{R_{11}} = 9\frac{V}{V}$$

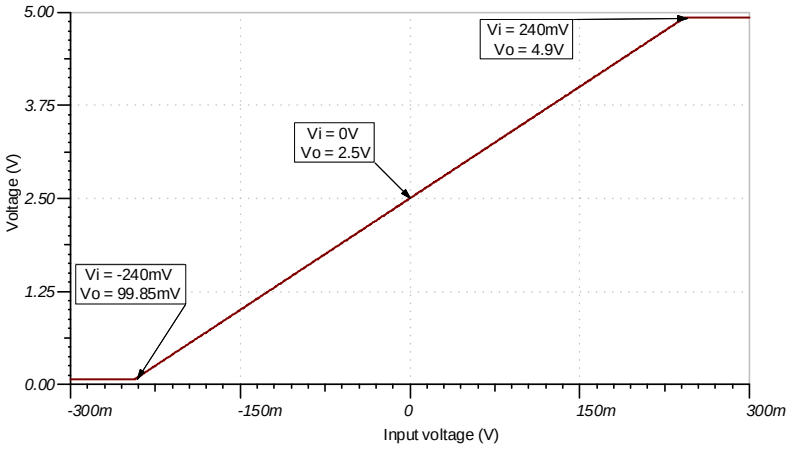
$$R_{11} = \frac{1000\,\Omega}{9\,\frac{V}{V}} = 111.11\,\Omega \rightarrow R_{11} = 111\ \Omega \quad \text{(Standard value)}$$

5. Calculate the reference voltage to bias the input to mid-supply. This will maximize the linear output swing of the instrumentation amplifier. See References for more information on the linear operating region of instrumentation amplifiers.

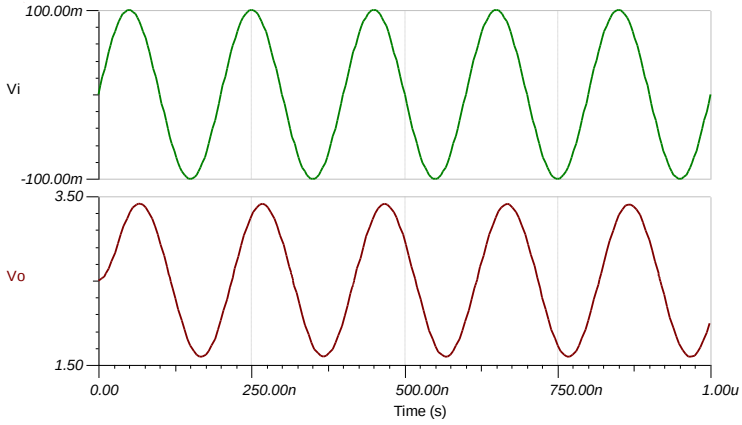
$$V_{ref} = \frac{V_s}{2} = \frac{5\ V}{2} = 2.5\ V$$

Design Simulations

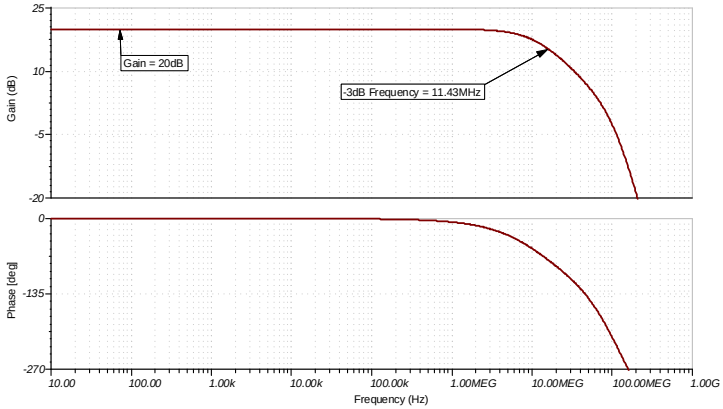
DC Simulation Results



Transient Simulation Results



AC Simulation Results



References

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAU6
3. TI Precision Labs
4. Instrumentation Amplifier V_{CM} vs. V_{OUT} Plots
5. Common-mode Range Calculator for Instrumentation Amplifiers

Design Featured Op Amp

OPA354	
V_{ss}	2.5V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	2mV
I_q	4.9mA/Ch
I_b	3pA
UGBW	250MHz
SR	150V/µs
#Channels	1,2,4
www.ti.com/product/opa354	

Design Alternate Op Amp

OPA322	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	500µV
I_q	1.6mA/Ch
I_b	0.2pA
UGBW	20MHz
SR	10V/µs
#Channels	1,2,4
www.ti.com/product/opa322	

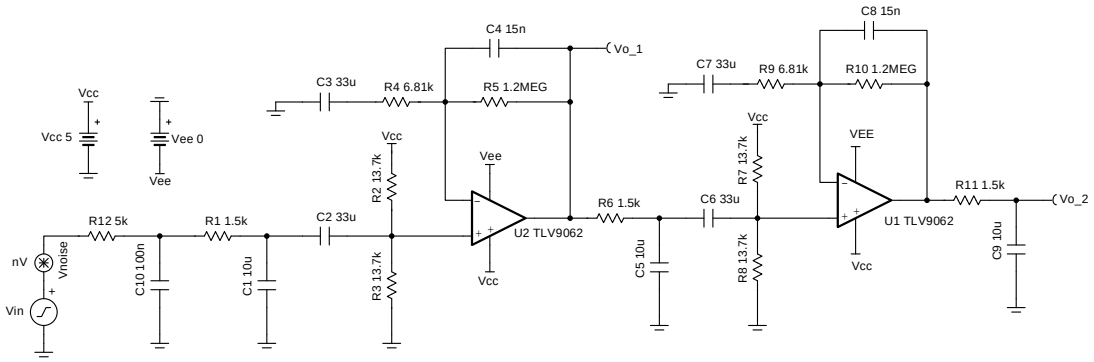
Low-noise and long-range PIR sensor conditioner circuit

Design Goals

AC Gain	Filter Cut Off Frequency		Supply	
90dB	f_L	f_H	V_{cc}	V_{ee}
	0.7Hz	10Hz	5V	0V

Design Description

This two stage amplifier design amplifies and filters the signal from a passive infrared (PIR) sensor. The circuit includes multiple low–pass and high–pass filters to reduce noise at the output of the circuit to be able to detect motion at long distances and reduce false triggers. This circuit can be followed by a window comparator circuit to create a digital output or connect directly to an analog–to–digital converter (ADC) input.



Design Notes

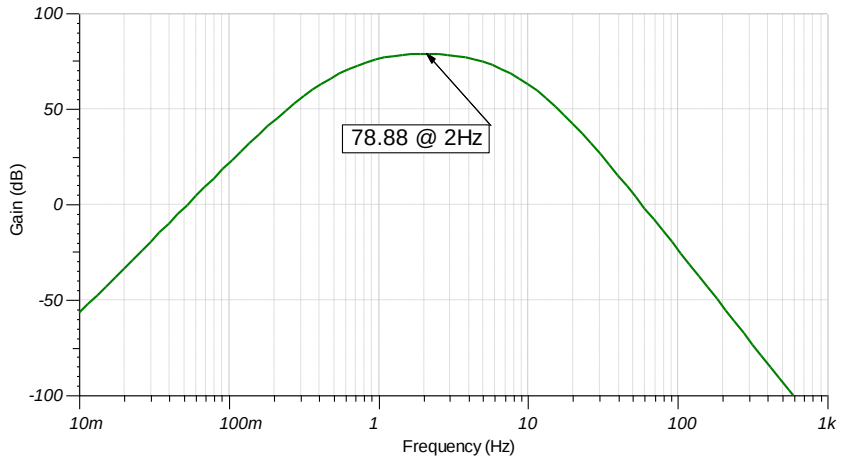
1. The common mode voltage and output bias voltage are set using the resistor dividers between R_2 and R_3 (and R_7 and R_8).
2. Two or more amplifier stages must be used to allow for sufficient loop gain.
3. Additional low–pass and high–pass filters can be added to further reduce noise.
4. Capacitors C_4 and C_8 filter noise by decreasing the bandwidth of the circuit and help stabilize the amplifiers.
5. RC filters on the output of the amplifiers (for example, R_6 and C_5) are required to reduce the total integrated noise of the amplifier.
6. The maximum gain of the circuit can be affected by the cutoff frequencies of the filters. The cutoff frequencies may need to be adjusted to achieve the desired gain.

Design Steps

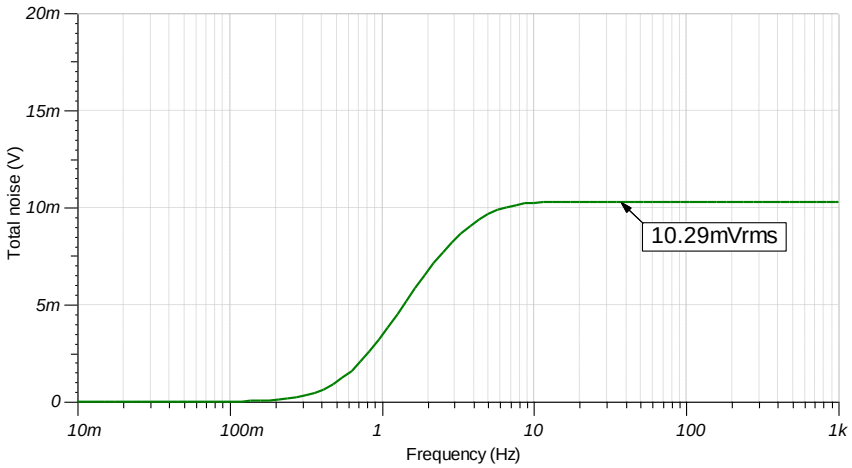
1. Choose large–valued capacitors C_1, C_5, and C_9 for the low–pass filters. These capacitors should be selected first since large–valued capacitors have limited standard values to select from compared to standard resistor values.
   $C_1 = C_5 = C_9 = 10\mu F$
2. Calculate resistor values for R_1, R_6, and R_{11} to form the low–pass filters.
   $R_1 = R_6 = R_{11} = \frac{1}{2\pi \times f_L \times C_1} = \frac{1}{2\pi \times 0.7Hz \times 10\mu F} = 1.592k\Omega$
   Choose $R_1 = R_6 = R_{11} = 1.5k\Omega$ (Standard value)
3. Select capacitor values for C_2, C_3, C_6, and C_7 for the high–pass filters.
   $C_2 = C_3 = C_6 = C_7 = 33\mu F$
4. Calculate the resistor values for R_4 and R_9 for the high–pass filters.
   $R_4 = R_9 = \frac{1}{2\pi \times f_H \times C_2} = \frac{1}{2\pi \times 10Hz \times 33\mu F} = 6.89k\Omega$
   Choose $R_4 = R_9 = 6.81k\Omega$ (Standard value)
5. Set the common–mode voltage of the amplifier to mid–supply using a voltage divider. The equivalent resistance of the voltage divider should be equal to R_4 to properly set the corner frequency of the high–pass filter.
   $R_2 = R_3 = R_7 = R_8 = 2 \times R_4 = 2 \times 6.81k\Omega = 13.62k\Omega$
   Choose $R_2 = R_3 = R_7 = R_8 = 13.7k\Omega$ (Standard value)
6. Calculate the gain required by each gain stage to achieve the total gain requirement. Distribute the total gain target of the circuit evenly between both gain stages.
   $Gain = \frac{90dB}{2} = 45dB = 177.828\frac{V}{V}$
7. Calculate R_5 to set the gain of the first stage.
   $R_5 = (Gain - 1) \times R_4 = (177.828\frac{V}{V} - 1) \times 6.81k\Omega = 1.204M\Omega$
   Choose $1.2M\Omega$
8. Calculate C_4 to set the low–pass filter cut off frequency.
   $C_4 = \frac{1}{2\pi \times R_5 \times f_L} = \frac{1}{2\pi \times 1.2MHz \times 10Hz} = 13.263nF$
   Choose $C_4 = 15nF$
9. Since the gain and cut off frequency of the first gain stage is equal to the second gain stage, set all component values of both stages equal to each other.
   $R_1 = R_6 = 5k\Omega$
   $R_7 = R_8 = 13.7k\Omega$
   $R_9 = R_4 = 6.81k\Omega$
   $R_{10} = R_5 = 1.2M\Omega$
   $C_8 = C_4 = 15nF$
10. Calculate R_{11} to set the cut off frequency of the low–pass filter at the output of the circuit.
    $R_{11} = \frac{1}{2\pi \times C_9 \times f_L} = \frac{1}{2\pi \times 10\mu F \times 10Hz} = 1.592k\Omega$
    Choose $R_{11} = 1.5k\Omega$

Design Simulations

AC Simulation Results



Noise Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOC524
3. TI Precision Labs

Design Featured Op Amp

TLV9062	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	0.3mV
I_q	538µA
I_b	0.5pA
UGBW	10MHz
SR	6.5V/µs
#Channels	1,2,4
www.ti.com/product/tlv9062	

Design Alternate Op Amp

OPA376	
V_{ss}	2.2V to 5.5V
V_{inCM}	V_{ee} to V_{cc}–1.3V
V_{out}	Rail–to–rail
V_{os}	5µV
I_q	760µA/Ch
I_b	0.2pA
UGBW	5.5MHz
SR	2V/µs
#Channels	1, 2, 4
http://www.ti.com/product/opa376	

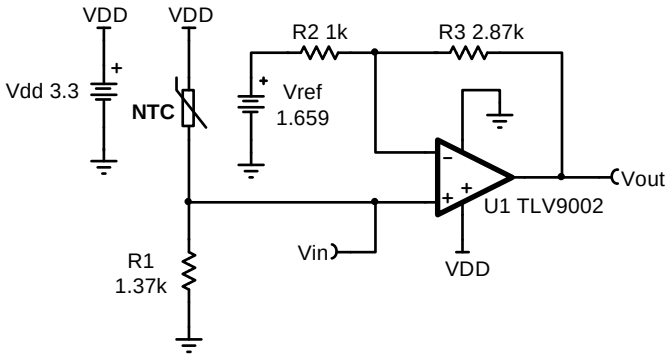
Temperature sensing with NTC circuit

Design Goals

Temperature		Output Voltage		Supply		
T_{Min}	T_{Max}	V_{outMin}	V_{outMax}	V_{dd}	V_{ee}	V_{ref}
25 ℃	50 ℃	0.05V	3.25V	3.3V	0V	1.659V

Design Description

This temperature sensing circuit uses a resistor in series with a negative–temperature–coefficient (NTC) thermistor to form a voltage divider, which has the effect of producing an output voltage that is linear over temperature. The circuit uses an op amp in a non–inverting configuration with inverting reference to offset and gain the signal, which helps to utilize the full ADC resolution and increase measurement accuracy.



Design Notes

1. Use the op amp in a linear operating region. Linear output swing is usually specified under the A_{OL} test conditions.
2. The connection, Vin, is a positive temperature coefficient output voltage. To correct an NTC output voltage, switch the position of R_1 and the NTC thermistor.
3. R_1 is chosen based on the temperature range and the NTC's value.
4. V_{ref} can be created using a DAC or voltage divider. If a voltage divider is used the equivalent resistance of the voltage divider will influence the gain of the circuit.
5. Using high value resistors can degrade the phase margin of the amplifier and introduce additional noise in the circuit. It is recommended to use resistor values around 10 kΩ or less.
6. A capacitor placed in parallel with the feedback resistor will limit bandwidth, improve stability and help reduce noise.

Design Steps

$$V_{out} = V_{dd} \times \frac{R_1}{R_{NTC} + R_1} \times \frac{R_2 + R_3}{R_2} - \frac{R_3}{R_2} \times V_{ref}$$

1. Calculate the value of R_1 to produce a linear output voltage. Use the minimum and maximum values of the PTC to obtain a range of values for R_1.

   $$R_{NTC\_max} = R_{NTC\ @25℃} = 2.252\ k\Omega, \quad R_{NTC\_min} = R_{NTC\ @50℃} = 819.7\ \Omega$$

   $$R_1 = \sqrt{R_{NTC\ @25℃} \times R_{NTC\ @50℃}} = \sqrt{2.252\ k\Omega \times 819.7\ \ \Omega} = 1.359\ k\Omega \approx 1.37\ k\Omega$$

2. Calculate the input voltage range.

   $$V_{inMin} = V_{dd} \times \frac{R_1}{R_{NTC\_max} + R_1} = 3.3\ V \times \frac{1.37\ k\Omega}{2.252\ k\Omega + 1.37\ k\Omega} = 1.248\ V$$

   $$V_{inMax} = V_{dd} \times \frac{R_1}{R_{NTC\_min} + R_1} = 3.3\ V \times \frac{1.37\ k\Omega}{819.7\ \Omega + 1.37\ k\Omega} = 2.065\ V$$

3. Calculate the gain required to produce the maximum output swing.

   $$G_{ideal} = \frac{V_{outMax} - V_{outMin}}{V_{inMax} - V_{inMin}} = \frac{3.25\ V - 0.05\ V}{2.065\ V - 1.248\ V} = 3.917\ \frac{V}{V}$$

4. Select R_2 and calculate R_3 to set the gain in Step 3.

   $$Gain = \frac{R_2 + R_3}{R_2}$$

   $$R_2 = 1\ k\Omega\ \ (\text{Standard value})$$

   $$R_3 = R_2 \times (G_{ideal} - 1) = 1\ k\Omega \times (3.917\frac{V}{V} - 1) = 2.917\ k\Omega$$

   $$\text{Choose}\ \ R_3 = 2.87\ k\Omega\ \ (\text{Standard value})$$

5. Calculate the actual gain based on standard values of R_2 and R_3.

   $$G_{actual} = \frac{R_2 + R_3}{R_2} = \frac{1\ k\Omega + 2.87\ k\Omega}{1\ k\Omega} = 3.87\ \frac{V}{V}$$

6. Calculate the output voltage swing based on the actual gain.

   $$V_{out\_swing} = (V_{inMax} - V_{inMin}) \times G_{actual} = (2.065\ V - 1.248\ V) \times 3.87\ \frac{V}{V} = 3.162\ V$$

7. Calculate the maximum output voltage when the output voltage is symmetrical around mid–supply.

   $$V_{outMax} = V_{mid-supply} + \frac{V_{out\_swing}}{2} = \frac{V_{dd} - V_{ee}}{2} + \frac{V_{out\_swing}}{2} = \frac{3.3\ V - 0\ V}{2} + \frac{3.162\ V}{2} = 3.231\ \ V$$

8. Calculate the reference voltage.

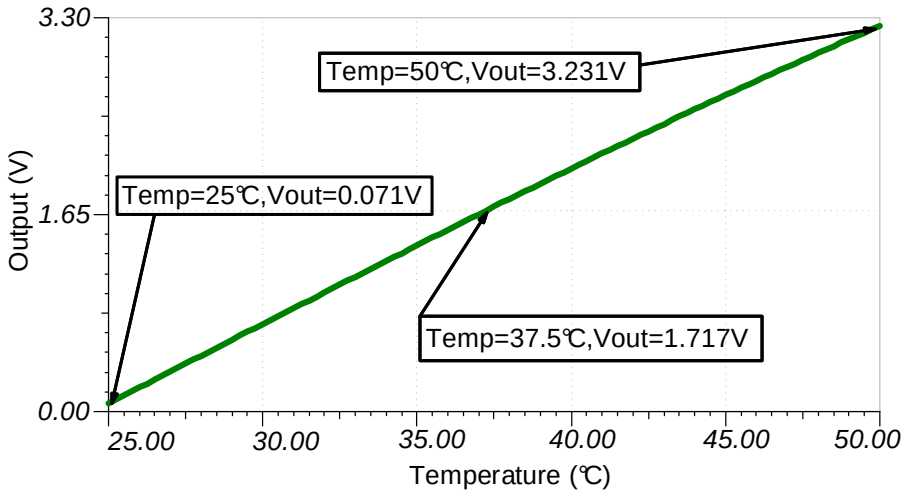
   $$V_{outMax} = V_{inMax} \times G_{actual} - \frac{R_3}{R_2} \times V_{ref}$$

   $$3.231\ V = 2.065\ V \times 3.87\ \frac{V}{V} - \frac{2.87\ k\Omega}{1\ k\Omega} \times V_{ref}$$

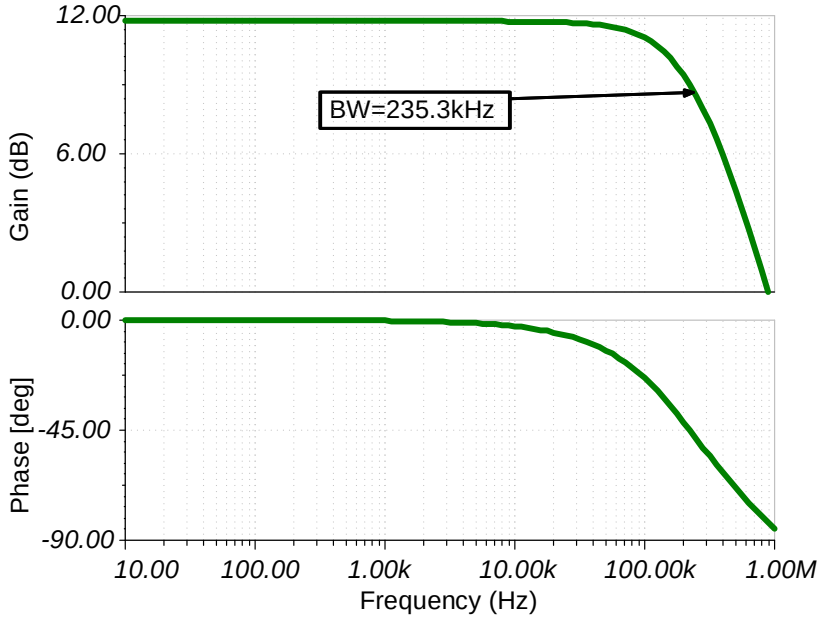
   $$V_{ref} = \frac{2.065\ V \times 3.87\ \frac{V}{V} - 3.231\ V}{\frac{2.87\ k\Omega}{1\ k\Omega}} = 1.659\ V$$

Design Simulations

DC Transfer Results



AC Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAV6
3. TI Precision Labs

Design Featured Op Amp:

TLV9002	
V_{cc}	1.8 V to 5.5 V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	1.5mV
I_q	0.06mA
I_b	5pA
UGBW	1MHz
SR	2V/µs
#Channels	1, 2, 4
http://www.ti.com/product/TLV9002	

Design Alternate Op Amp:

OPA333	
V_{cc}	1.8 V to 5.5 V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	2µV
I_q	17µA
I_b	70pA
UGBW	350kHz
SR	0.16V/µs
#Channels	1, 2, 4
http://www.ti.com/product/OPA333	

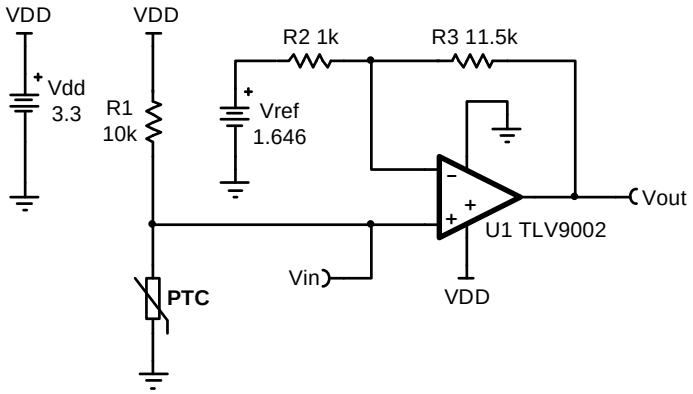
Temperature sensing with PTC circuit

Design Goals

Temperature		Output voltage		Supply		
T_{Min}	T_{Max}	V_{outMin}	V_{outMax}	V_{dd}	V_{ee}	V_{ref}
0 ℃	50 ℃	0.05V	3.25V	3.3V	0V	1.646V

Design Description

This temperature sensing circuit uses a resistor in series with a positive–temperature–coefficient (PTC) thermistor to form a voltage–divider, which has the effect of producing an output voltage that is linear over temperature. The circuit uses an op amp in a non–inverting configuration with inverting reference to offset and amplify the signal, which helps to utilize the full ADC resolution and increase measurement accuracy.



Design Notes

1. Use the op amp in a linear operating region. Linear output swing is usually specified under the A_{OL} test conditions.
2. The connection, V_{in}, is a positive temperature coefficient output voltage. To correct a negative–temperature–coefficient (NTC) output voltage, switch the position of R_1 and PTC resistor.
3. Choose R_1 based on the temperature range and the PTC's value.
4. V_{ref} can be created using a DAC or voltage divider. If a voltage divider is used the equivalent resistance of the voltage divider will alter the gain of the circuit and should be accounted for.
5. Using high–value resistors can degrade the phase margin of the amplifier and introduce additional noise in the circuit. It is recommended to use resistor values around 10kΩ or less.
6. A capacitor placed in parallel with the feedback resistor will limit bandwidth, improve stability and help reduce noise.

Design Steps

$$V_{out} = V_{dd} \times \frac{R_{PTC}}{R_{PTC} + R_1} \times \frac{R_2 + R_3}{R_2} - \frac{R_3}{R_2} \times V_{ref}$$

1. Calculate the value of R_1 to produce a linear output voltage. Use the minimum and maximum values of the PTC to obtain a range of values for R_1.

$$R_{PTC\_Max} = R_{PTC\ @50℃} = 11.611\ k\Omega$$

$$R_{PTC\_Min} = R_{PTC\ @0℃} = 8.525\ k\Omega$$

$$R_1 = \sqrt{R_{PTC\ @0℃} \times R_{PTC\ @50℃}} = \sqrt{8.525\ k\Omega \times 11.611\ k\Omega} = 9.95\ k\Omega \approx 10\ k\Omega$$

2. Calculate the input voltage range.

$$V_{inMin} = V_{dd} \times \frac{R_{PTC\_Min}}{R_{PTC\_Min} + R_1} = 3.3\ V \times \frac{8.525\ k\Omega}{8.525\ k\Omega + 10\ k\Omega} = 1.519\ V$$

$$V_{inMax} = V_{dd} \times \frac{R_{PTC\_Max}}{R_{PTC\_Max} + R_1} = 3.3\ V \times \frac{11.611\ k\Omega}{11.611\ k\Omega + 10\ k\Omega} = 1.773\ V$$

3. Calculate the gain required to produce the maximum output swing.

$$G_{ideal} = \frac{V_{outMax} - V_{outMin}}{V_{inMax} - V_{inMin}} = \frac{3.25\ V - 0.05\ V}{1.773\ V - 1.519\ V} = 12.598\ \frac{V}{V}$$

4. Select R_2 and calculate R_3 to set the gain calculated in Step 3.

$$Gain = \frac{R_2 + R_3}{R_2}$$

$$R_2 = 1\ k\Omega$$

$$R_3 = R_2 \times (G_{ideal} - 1) = 1\ k\Omega \times (12.598 - 1) = 11.598\ k\Omega$$

$$\text{Choose}\quad R_3 = 11.5\ k\Omega\ \ (\textit{Standard value})$$

5. Calculate the actual gain based on standard values of R_2 and R_3.

$$G_{actual} = \frac{R_2 + R_3}{R_2} = \frac{1\ k\Omega + 11.5\ k\Omega}{1\ k\Omega} = 12.5\ \frac{V}{V}$$

6. Calculate the output voltage swing based on the actual gain.

$$V_{out\_swing} = (V_{inMax} - V_{inMin}) \times G_{actual} = (1.773\ V - 1.519\ V) \times 12.5\ \frac{V}{V} = 3.175\ V$$

7. Calculate the maximum output voltage when the output voltage is symmetrical around mid–supply.

$$V_{outMax} = V_{mid-supply} + \frac{V_{out\_swing}}{2} = \frac{V_{dd} - V_{ee}}{2} + \frac{V_{out\_swing}}{2} = \frac{3.3\ V - 0\ V}{2} + \frac{3.175\ V}{2} = 3.238\ V$$

8. Calculate the reference voltage.

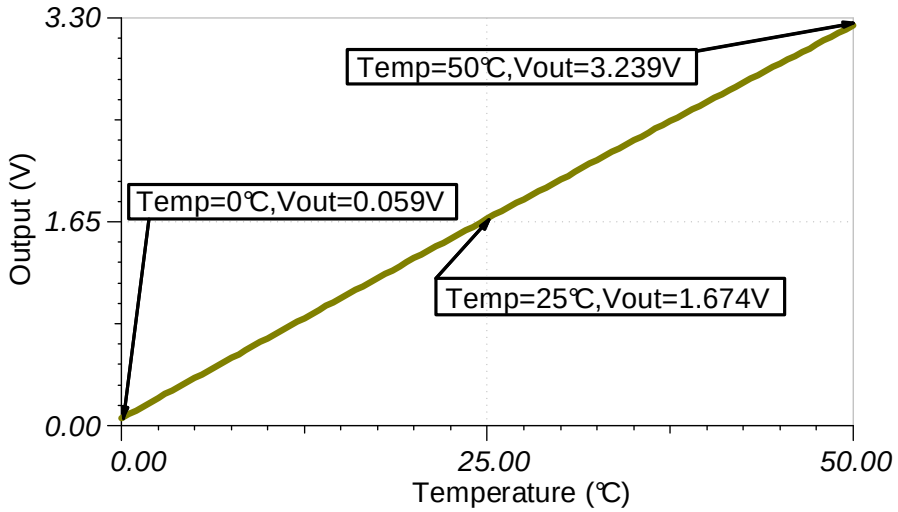
$$V_{outMax} = V_{inMax} \times G_{actual} - \frac{R_3}{R_2} \times V_{ref}$$

$$3.238\ V = 1.773\ V \times 12.5\ \frac{V}{V} - \frac{11.5\ k\Omega}{1\ k\Omega} \times V_{ref}$$

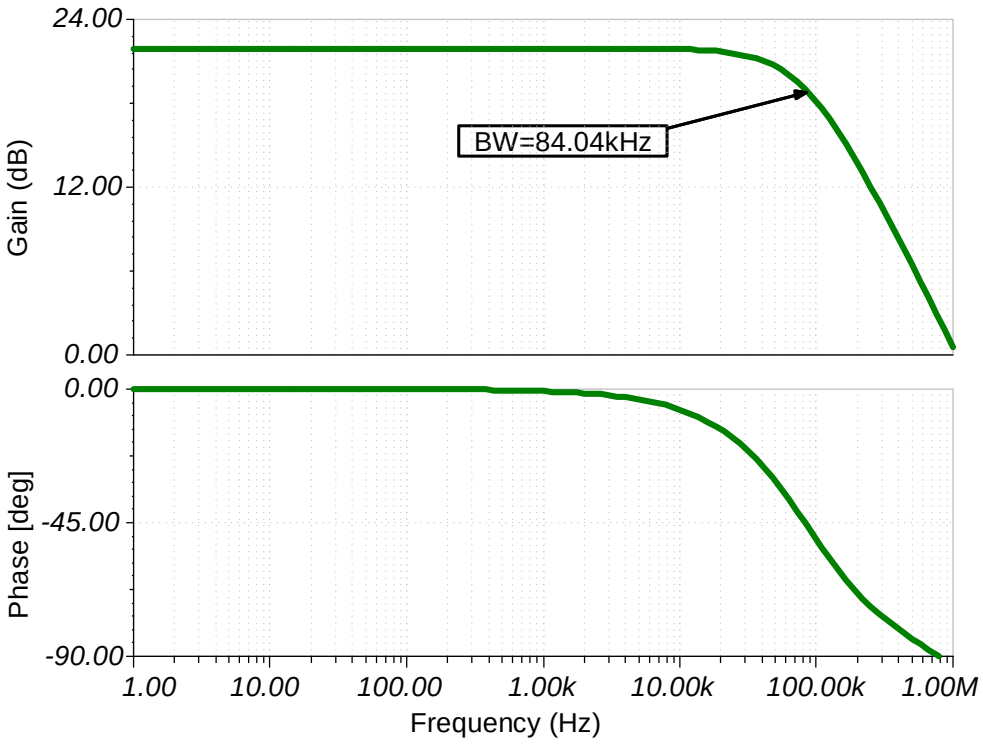
$$V_{ref} = \frac{1.773\ V \times 12.5\ \frac{V}{V} - 3.238\ V}{\frac{11.5\ k\Omega}{1\ k\Omega}} = 1.646\ V$$

Design Simulations

DC Transfer Results



AC Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAV5
3. TI Precision Labs

Design Featured Op Amp

TLV9002	
V_{cc}	1.8 V to 5.5 V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	1.5mV
I_q	0.06mA
I_b	5pA
UGBW	1MHz
SR	2V/µs
#Channels	1, 2, 4
http://www.ti.com/product/TLV9002	

Design Alternate Op Amp

OPA333	
V_{cc}	1.8 V to 5.5 V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	2µV
I_q	17µA
I_b	70pA
UGBW	350kHz
SR	0.16V/µs
#Channels	1, 2, 4
http://www.ti.com/product/OPA333	

Differential input to differential output circuit using a fully-differential amplifier

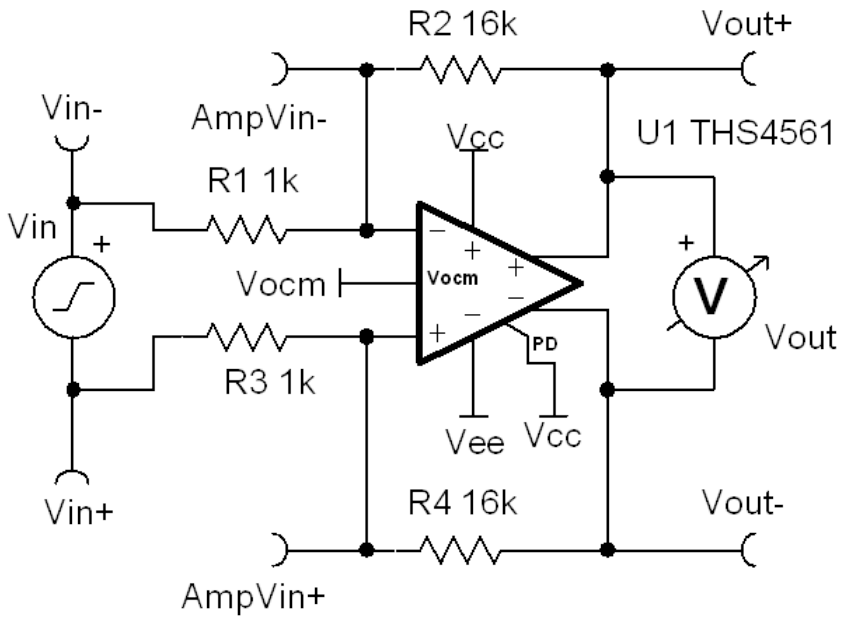
Design Goals

Input	Output	Supply	
Differential	Differential	V_{cc}	V_{ee}
1Vpp	16Vpp	10V	0V

Output Common-Mode	3dB Bandwidth	AC Gain (Gac)
5V	3MHz	16V/V

Design Description

This design uses a fully differential amplifier (FDA) as a differential input to differential output amplifier.



Design Notes

1. The ratio R2/R1, equal to R4/R3, sets the gain of the amplifier.
2. For a given supply, the output swing for and FDA is twice that of a single ended amplifier. This is because a fully differential amplifier swings both terminals of the output, instead of swinging one and fixing the other to either ground or a Vref. The minimum voltage of an FDA is therefore achieved when Vout+ is held at the negative rail and Vout- is held at the positive rail, and the maximum is achieved when Vout+ is held at the positive rail and Vout- is held at the negative rail.
3. FDAs are useful for noise sensitive signals, since noise coupling equally into both inputs will not be amplified, as is the case in a single ended signal referenced to ground.
4. The output voltages will be centered about the output common-mode voltage set by Vocm.
5. Both feedback paths should be kept symmetrical in layout.

Design Steps

- Set the ratio R2/ R1 to select the AC voltage gain. To keep the feedback paths balanced,

  $R_1 = R_3 = 1k\Omega\ (Standard\ Value)$

  $R_2 = R_4 = R_1 \bullet (G_{AC}) = 1k\Omega \bullet \left(16\frac{V}{V}\right) = 16k\Omega\ (Standard\ Value)$

- Given the output rails of 9.8V and 0.2V for Vs = 10V, verify that 16Vpp falls within the output range available for V_{ocm} = 5V.

  In normal operation:

  $Amp V_{IN+} = Amp V_{IN-}$

  $V_{OUT+} - V_{ocm} = V_{ocm} - V_{OUT-}$

  $V_{OUT} = V_{OUT+} - V_{OUT-}$

- Rearrange to solve for each output voltage in edge conditions

  $V_{OUT-} = 2V_{ocm} - V_{OUT+}$

  $V_{OUT-} = V_{OUT+} - V_{OUT}$

  $2V_{OUT+} = 2V_{ocm} + V_{OUT}$

  $V_{OUT+} = V_{ocm} + \frac{V_{OUT}}{2}$

  $V_{OUT-} = V_{ocm} - \frac{V_{OUT}}{2}$

- Verifying for Vout = +8V and Vocm = +5V,

  $V_{OUT+} = 5 + \frac{8}{2} = 9V < 9.8V$

  $V_{OUT-} = 5 - \frac{8}{2} = 1V > 0.2V$

- Verifying for Vout = -8V and Vocm = +5V,

  $V_{OUT+} = 5 + \frac{-8}{2} = 1V > 0.2V$

  $V_{OUT-} = 5 - \frac{-8}{2} = 9V > 9.8V$

Note that the maximum swing possible is:

$(9.8V - 0.2V) - (0.2V - 9.8V) = 18.4V_{pp}$, or $\pm 9.4V$

- Use the input common mode voltage range of the amplifier and the feedback resistor divider to find the signal input range when the output range is 1V to 9V. Due to symmetry, calculation of one side is sufficient.

$$Min(AmpV_{IN+}) = Min(AmpV_{IN-}) = Vee - 0.1V = -0.1V$$

$$Max(AmpV_{IN+}) = Max(AmpV_{IN-}) = Vcc - 1.1V = 8.9V$$

$$\frac{AmpV_{IN-} - V_{IN-}}{R_1} = \frac{V_{OUT+} - AmpV_{IN-}}{R_2}$$

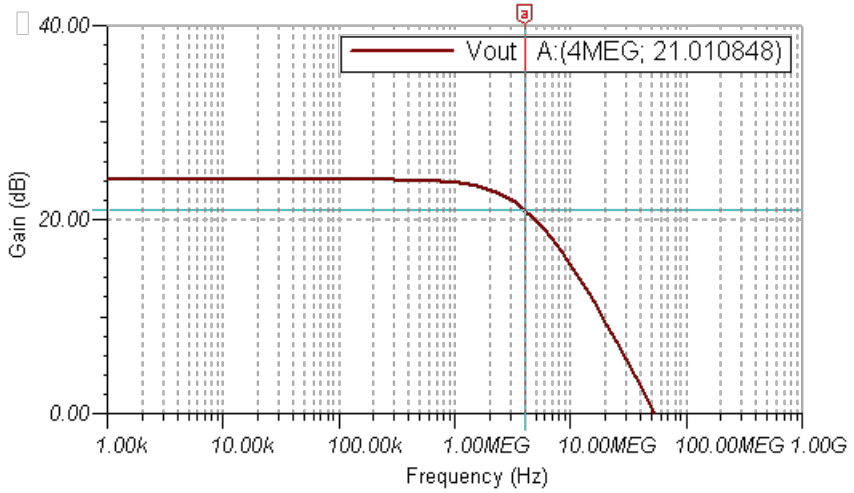
$$V_{IN-} = AmpV_{IN-} - \frac{V_{OUT+} - AmpV_{IN-}}{\frac{R_2}{R_1}}$$

$$Min(V_{IN-}) = -0.1V - \frac{9V - (-0.1V)}{16\ \frac{V}{V}} = -0.65V$$

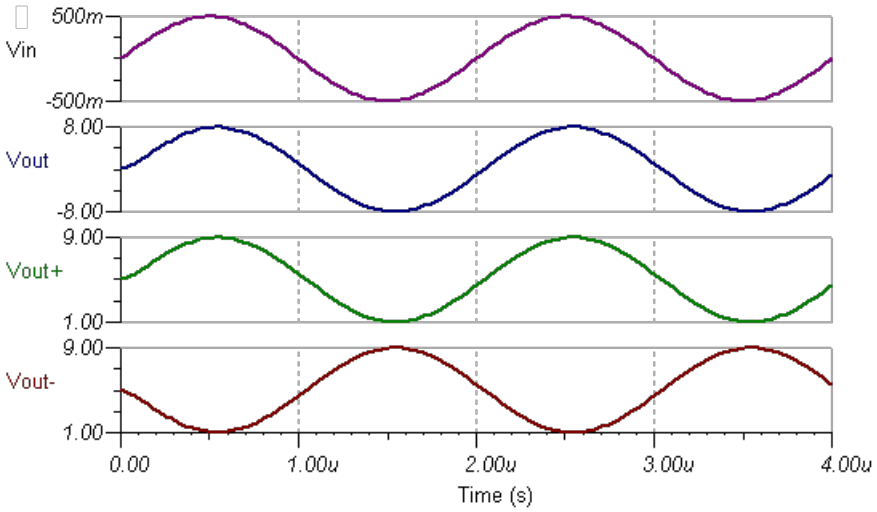
$$Max(V_{IN-}) = 8.9V + \frac{8.9V - 1V}{16\frac{V}{V}} = 9.4V$$

Design Simulations

AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the TIDA-01036 tool folder for more information.

Design Featured Op Amp

THS4561	
V_{ss}	3V to 13.5V
V_{inCM}	Vee-0.1V to Vcc-1.1V
V_{out}	Vee+0.2V to Vcc-0.2
V_{os}	TBD
I_q	TBD
I_b	TBD
UGBW	70MHz
SR	4.4V/µs
#Channels	1
http://www.ti.com/product/THS4561	

Design Alternate Op Amp

THS4131	
V_{ss}	5V to 33V
V_{inCM}	Vee+1.3V to Vcc-0.1V
V_{out}	Varies
V_{os}	2mV
I_q	14mA
I_b	2uA
UGBW	80MHz
SR	52V/µs
#Channels	1
http://www.ti.com/product/THS4131	

Single-ended input to differential output circuit using a fully-differential amplifier

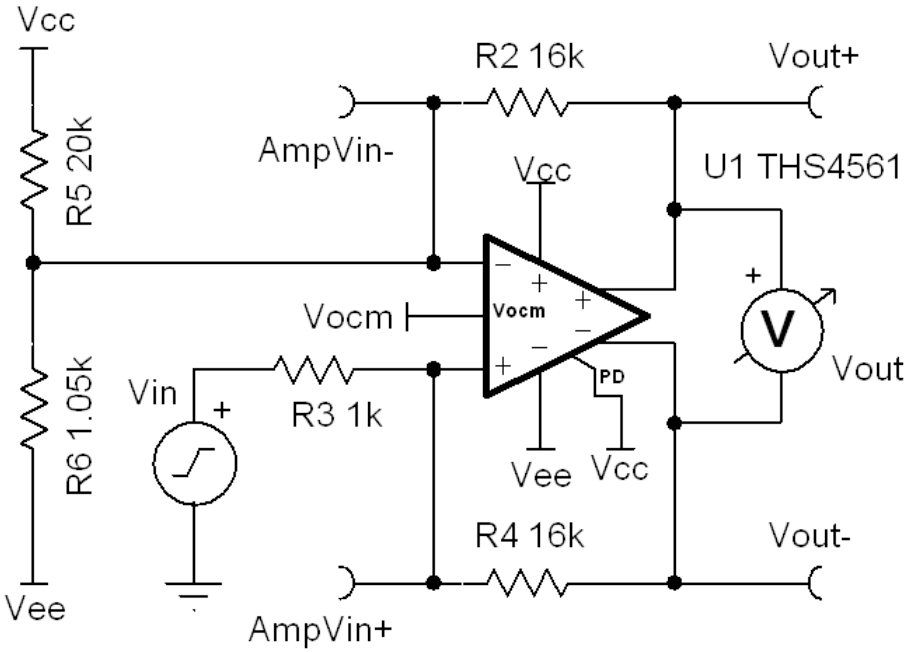
Design Goals

Input	Output	Supply	
Single-Ended	Differential	V_{cc}	V_{ee}
0V to 1V	16Vpp	10V	0V

Output Common-Mode	3dB Bandwidth	AC Gain (Gac)
5V	3MHz	16V/V

Design Description

This design uses a fully-differential amplifier (FDA) as a single-ended input to differential output amplifier.



Design Notes

1. The ratio R_4/R_3, equal to $R_2/(R_5||R_6)$, sets the gain of the amplifier.
2. The main difference between a single-ended input and a differential input is that the available input swing is only half. This is because one of the input voltages is fixed at a reference.
3. It is recommended to set this reference to mid-input signal range, rather than the min-input, to induce polarity reversal in the measured differential input. This preserves the ability of the outputs to crossover, which provides the doubling of output swing possible with an FDA.
4. The impedance of the reference voltage must be equal to the signal input resistor. This can be done by creating a resistor divider with a Thevnin equivalent of the correct reference voltage and impedance.

Design Steps

- Find the resistor divider with that produces a 0.5V, 1-kΩ reference from Vs = 10V.

$$\frac{R_6}{R_5 + R_6} = F \quad \frac{0.5V}{10V} \qquad \frac{R_5 \cdot R_6}{R_5 + R_6} \quad E = 1k\Omega$$

$$R_6 = FR_5 + FR_6$$

$$R_6(1-F) = FR_5$$

$$R_5 \quad \frac{R_6(1-F)}{F}$$

$$\frac{R_6(1-F)/F \cdot R_6}{R_6(1-F)/F + R_6} \quad E$$

$$\frac{{R_6}^2 \cdot (1-F)/F}{(R_6/F - R_6) + R_6} \quad E$$

$$\frac{{R_6}^2 \cdot (1-F)/F}{R_6/F} \quad E$$

$$R_6 \cdot (1-F) \quad E$$

$$R_6 \quad \frac{E}{1-F} \quad \frac{1k\Omega}{1-0.05} \quad 1.05k\Omega$$

$$R_5 \quad \frac{1.05\Omega(1-0.05)}{0.05} \quad 20k\Omega$$

- Verify that the minimum input of 0V and the maximum input of 1-V result in an output within the 9.4-V range available for Vocm = 5V.

  Since the resistor divider acts like a 0.5V reference, the measured differential input for a 0-V V_{IN} is:

  $V_{IN} = 0V - 0.5V = -0.5V$

- The output is:

$$-0.5V \cdot \frac{16V}{V} \quad -8V > -9.8V$$

- Likewise, for a 1-V input:

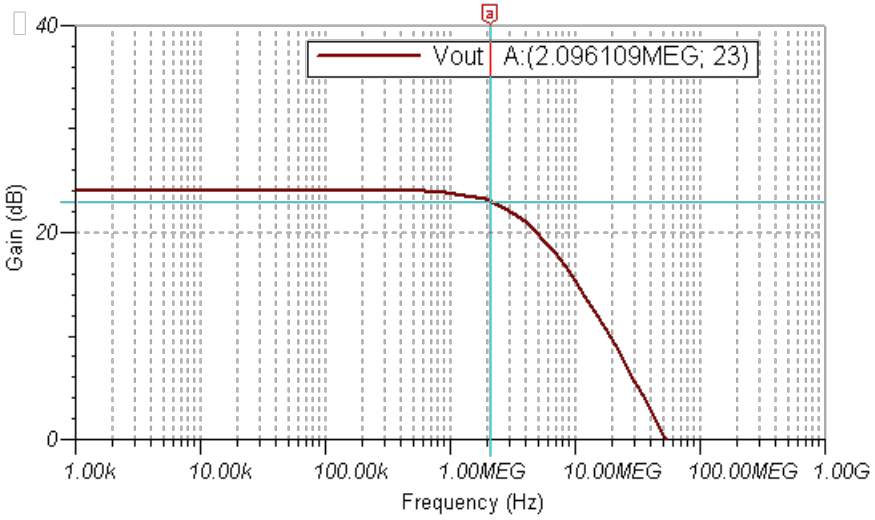
  $V_{IN} = 1V - 0.5V = 0.5V$

$$0.5V \cdot \frac{16V}{V} \quad 8V < 9.8V$$

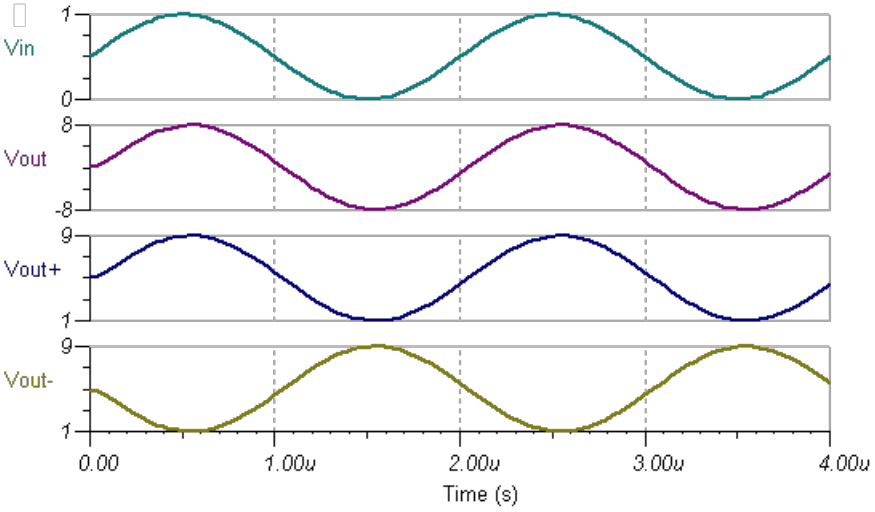
NOTE: With a reference voltage of 0V, a 1-V input results in an output voltage greater than the maximum output range of the amplifier.

Design Simulations

AC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the TI Precision Labs video – Op Amps: Fully Differential Amplifiers – Designing a Front-End Circuit for Driving a Differential Input ADC, for more information.

Design Featured Op Amp

THS4561	
V_{ss}	3V to 13.5V
V_{inCM}	Vee-0.1V to Vcc-1.1V
V_{out}	Vee+0.2V to Vcc-0.2
V_{os}	TBD
I_q	TBD
I_b	TBD
UGBW	70MHz
SR	4.4V/µs
#Channels	1
http://www.ti.com/product/THS4561	

Design Alternate Op Amp

THS4131	
V_{ss}	5V to 33V
V_{inCM}	Vee+1.3V to Vcc-0.1V
V_{out}	Varies
V_{os}	2mV
I_q	14mA
I_b	2uA
UGBW	80MHz
SR	52V/µs
#Channels	1
http://www.ti.com/product/THS4131	

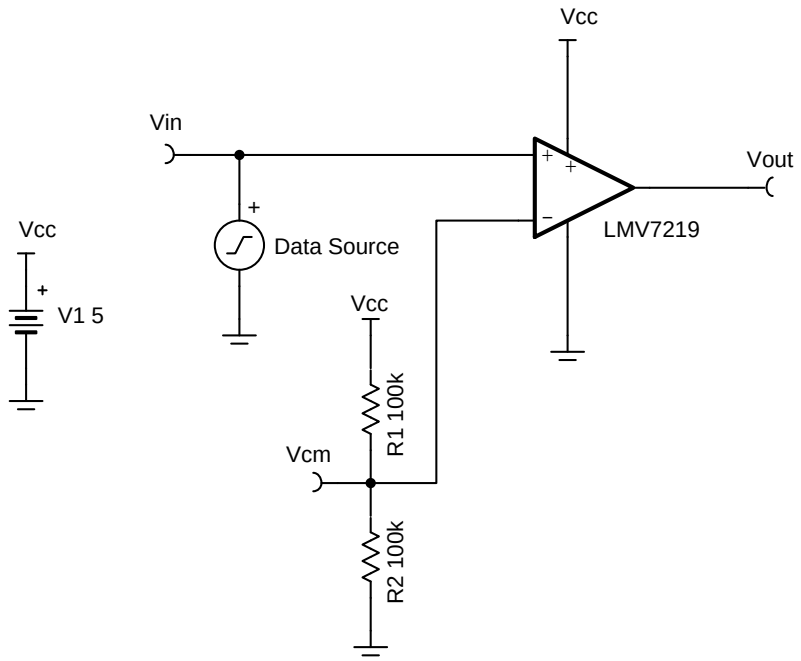
Signal and clock restoration circuit

Design Goals

Supply		Attenuated Input Signal		
V_{cc}	V_{ee}	V_i	V_{cm}	f
5V	0V	200mV_{p-p}	2.5V	20MHz

Design Description

The signal restoration circuit is used in digital systems to retrieve distorted clock or data waveforms. These clock and data signals can be attenuated and distorted on long traces due to stray capacitance, stray inductance, or reflections on transmission lines.The comparator is used to sense the attenuated and distorted input signal and convert it to a full scale digital output signal.



Design Notes

1. Select a comparator with low input offset voltage and fast propagation delay.
2. A comparator with a toggle frequency larger than the input signal frequency should be used in order to properly process the incoming digital signal. A margin of 30% is sufficient.
3. If level translation is also required, use a comparator with seperate input and output supplies.
4. If a differential output is required, use a comparator with a compatible output stage such as the LVDS compatible output on the LMH7220.

Design Steps

1. Calculate the maximum toggle frequency of the comparator to ensure it can process the 20MHz input signal.

   $f_{max} = (t_{rise} + t_{fall} + t_{pd\_hl} + t_{pd\_lh})^{-1}$

   $f_{max} = (1.3ns + 1.25ns + 7ns + 7ns)^{-1} = 35.4\ MHz$

2. Set the inverting input of the comparator to the common mode voltage of 2.5V through the resistor divider R_1, R_2.

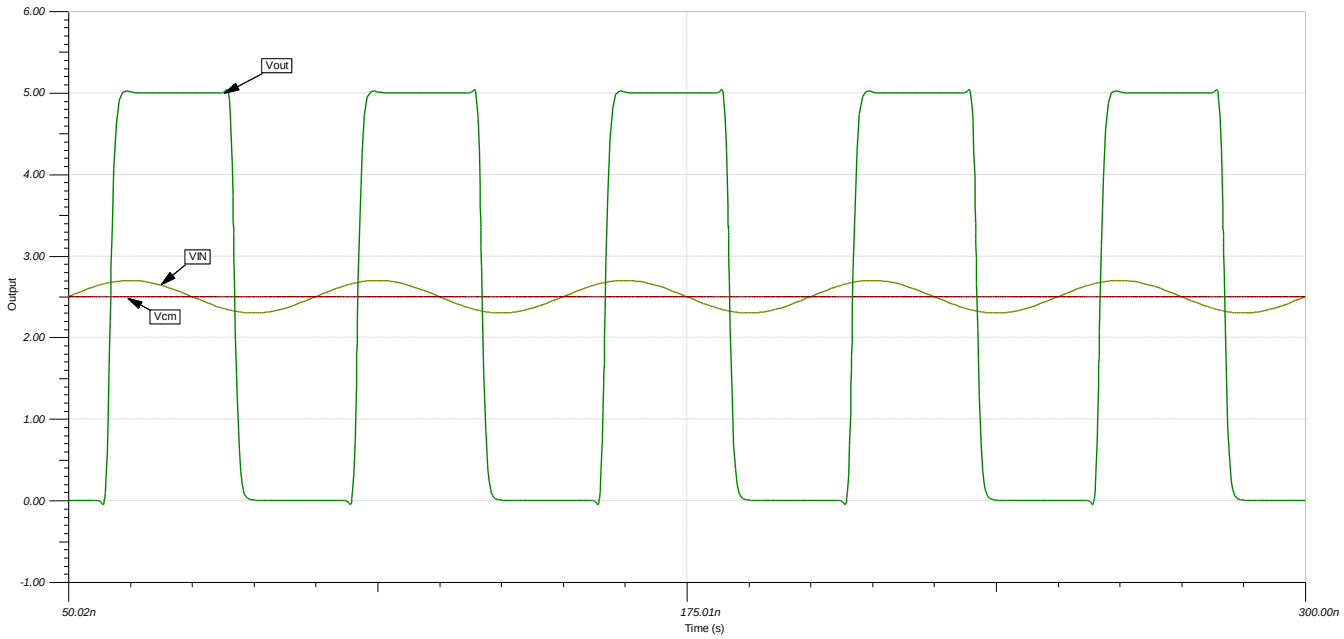
   $V_{cm} = (V_{cc})\ x\ (\frac{R_2}{R_1+R_2}) = 2.5V$

   $(\frac{R_2}{R_1+R_2}) = \frac{1}{2}$

3. Set $R_1 = R_2 = 100k$.
4. Set the noninverting input of the comparator to the input data signal.

Design Simulations

Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit spice simulation file, SNOM661.

For more information on many comparator topics including hysteresis, propagation delay and input common mode range please see, TI Precision Labs.

Design Featured Comparator

LMV7219	
V_{ss}	2.7V to 5V
V_{inCM}	Rail-to-rai
t_{pd}	7ns
V_{os}	1mV
V_{HYS}	7mV
I_q	0.9mA
Output Type	Push-Pull
#Channels	1
www.ti.com/product/lmv7219	

Design Alternate Comparator

	TLV3501	LMH7220
V_{ss}	2.7 to 5.5V	2.7V to 12V
V_{inCM}	Rail-to-rail	Rail-to-rail
t_{pd}	4.5ns	2.9ns
V_{os}	1mV	9.5mV
V_{HYS}	6mV	na
I_q	3.2mA	6.8
Output Type	Push-Pull	LVDS
#Channels	1	1
	www.ti.com/product/tlv3501	www.ti.com/product/lmh7220

Comparator with and without hysteresis circuit

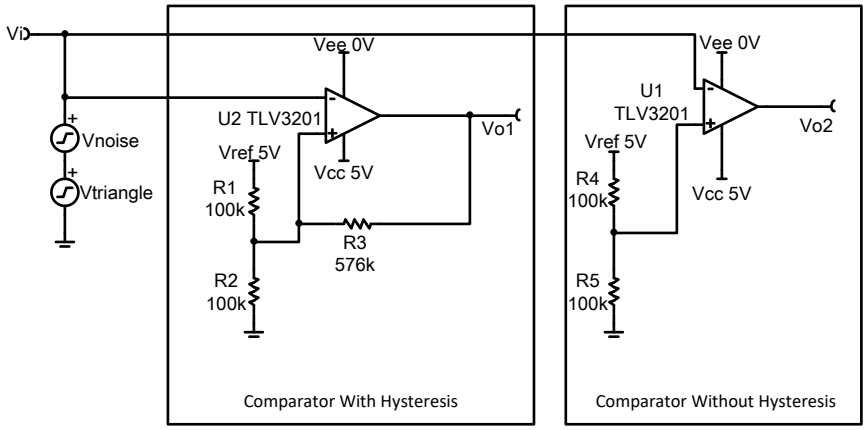
Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
0V	5V	0V	5V	5V	0V	5V

V_L (Lower Threshold)	V_H (Upper Threshold)	$V_H - V_L$
2.3V	2.7V	0.4V

Design Description

Comparators are used to compare two different signal levels and create an output based on the input with the higher input voltage. Noise or signal variation at the comparison threshold will cause the comparator output to have multiple output transitions. Hysteresis sets upper- and lower-threshold voltages to eliminate the multiple transitions caused by noise.



Design Notes

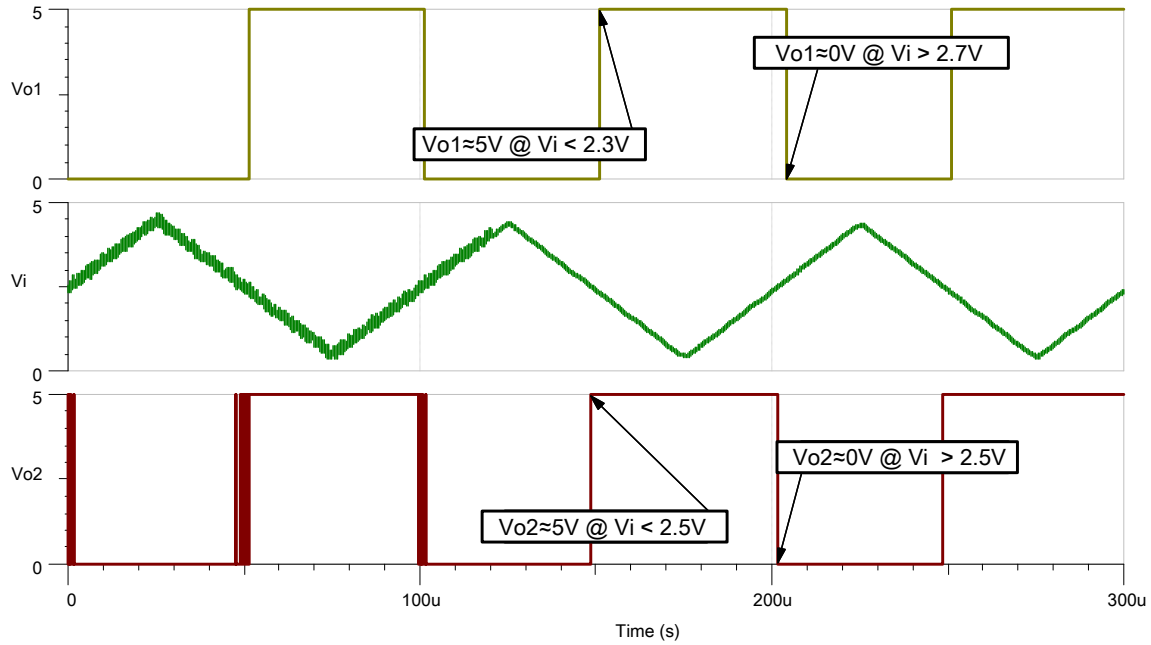
1. Use a comparator with low quiescent current to reduce power consumption.
2. The accuracy of the hysteresis threshold voltages are related to the tolerance of the resistors used in the circuit.
3. The propagation delay is based on the specifications of the selected comparator.

Design Steps

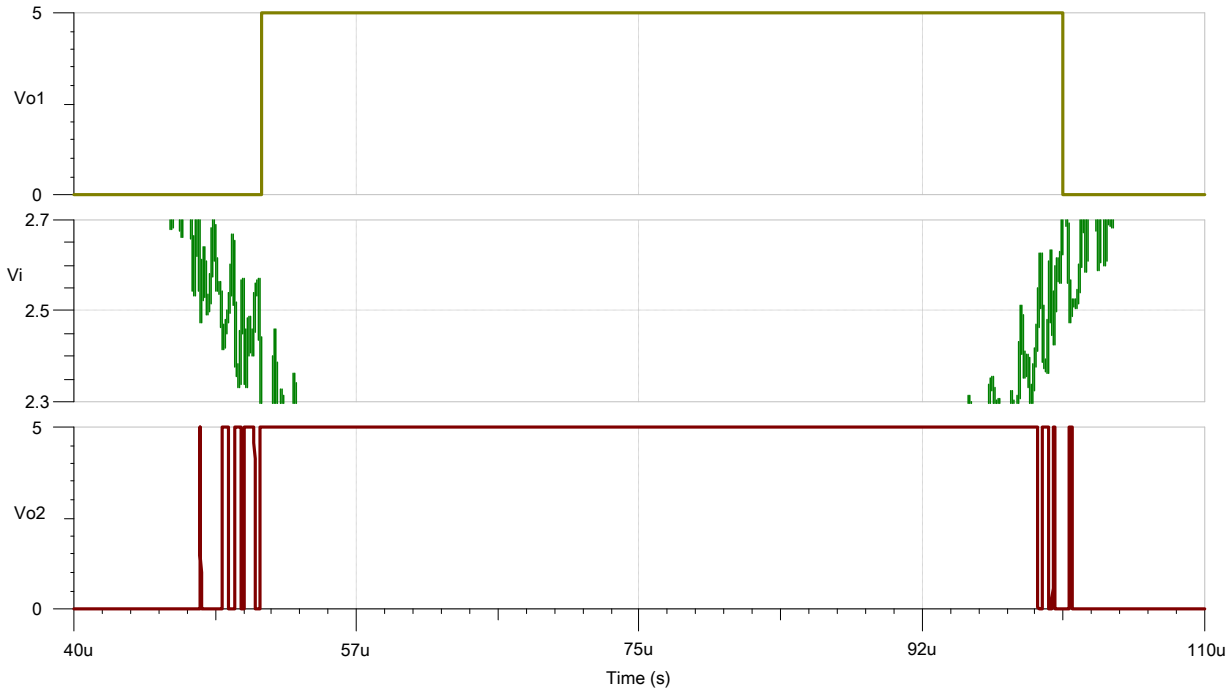
1. Select components for the comparator with hysteresis.
   a. Select V_L, V_H, and R_1.
      $V_L = 2.3V$
      $V_H = 2.7V$
      $R_1 = 100k\Omega$ (Standard Value)
   b. Calculate R_2.
      $R_2 = \frac{V_L}{V_{cc} - V_H} \times R_1 = \frac{2.3V}{5V - 2.7V} \times 100k\Omega = 100k\Omega$ (Standard Value)
   c. Calculate R_3.
      $R_3 = \frac{V_L}{V_H - V_L} \times R_1 = \frac{2.3V}{2.7V - 2.3V} \times 100k\Omega = 575k\Omega \approx 576k\Omega$ (Standard Value)
   d. Verify hysteresis width.
      $V_H - V_L = \frac{R_1 \times R_2}{(R_3 \times R_1) + (R_3 \times R_2) + (R_1 \times R_2)} \times V_{cc}$
      $= \frac{100k\Omega \times 100k\Omega}{(576k\Omega \times 100k\Omega) + (576k\Omega \times 100k\Omega) + (100k\Omega \times 100k\Omega)} \times 5V = 0.399V$
2. Select components for comparator without hysteresis.
   a. Select V_{th} and R_4 .
      $V_{th} = 2.5V$
      $R_4 = 100k\Omega$ (Standard Value)
   b. Calculate R_5.
      $R_5 = \frac{V_{th}}{V_{cc} - V_{th}} \times R_4 = \frac{2.5V}{5V - 2.5V} \times 100k\Omega = 100k\Omega$ (Standard Value)

Design Simulations

Transient Simulation Results



Noise Only Present From 0s to 120µs



Zoomed in From 40µs to 110µs

Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC515.

See TIPD144, www.ti.com/tool/tipd144.

Design Featured Comparator

TLV3201	
V_{cc}	2.7V to 5.5V
V_{inCM}	Extends 200mV beyond either rail
V_{out}	(V_{ee}+230mV) to (V_{cc}–210mV) @ 4mA
V_{os}	1mV
I_q	40µA
I_b	1pA
UGBW	-
SR	-
#Channels	1, 2
www.ti.com/product/tlv3201	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

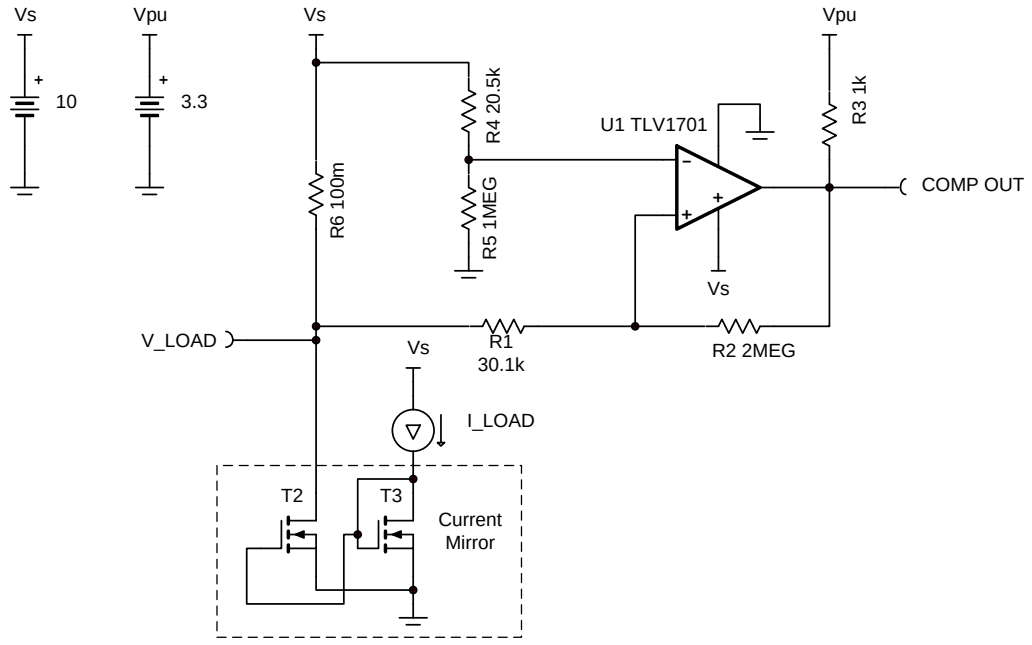
High-side current sensing with comparator circuit

Design Goals

Load Current (I_L)		System Supply (V_S)	Comparator Output Status	
Over Current (I_{OC})	Recovery Current (I_{RC})	Typical	Over Current	Normal Operation
1 A	0.5 A	10 V	V_{OL} < 0.4 V	V_{OH} = V_{PU} = 3.3 V

Design Description

This high-side, current sensing solution uses one comparator with a rail-to-rail input common mode range to create an over-current alert (OC-Alert) signal at the comparator output (COMP OUT) if the load current rises above 1A. The OC-Alert signal in this implementation is active low. So when the 1A threshold is exceeded, the comparator output goes low. Hysteresis is implemented such that OC-Alert will return to a logic high state when the load current reduces to 0.5A (a 50% reduction). This circuit utlizes an open-drain output comparator in order to level shift the output high logic level for controlling a digital logic input pin. For applications needing to drive the gate of a MOSFET switch, a comparator with a push-pull output is preferred.



Design Notes

1. Select a comparator with rail-to-rail input common mode range to enable high-side current sensing.
2. Select a comparator with an open-drain output stage for level-shifting.
3. Select a comparator with low input offset voltage to optimize accuracy.
4. Calculate the value for the shunt resistor (R_6) so the shunt voltage (V_{SHUNT}) is at least ten times larger than the comparator offset voltage (V_{IO}).

Design Steps

1. Select value of R_6 so V_{SHUNT} is at least 10x greater than the comparator input offset voltage (V_{IO}). Note that making R_6 very large will improve OC detection accuracy but will reduce supply headroom.

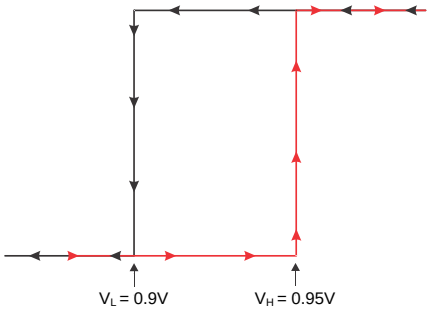
   $V_{SHUNT} = (I_{OC} \times R_6) \geq 10 \times V_{IO} = 55mV$

   set $R_6 = 100m\Omega$ for $I_{OC} = 1A$ & $V_{IO} = 5.5mV$

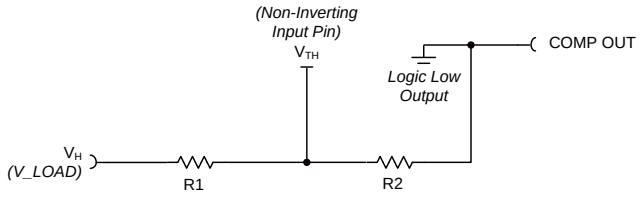
2. Determine the desired switching thresholds for when the comparator output will transition from high-to-low (V_L) and low-to-high (V_H). V_L represents the threshold when the load current crosses the OC level, while V_H represents the threshold when the load current recovers to a normal operating level.

   $V_L = V_S - (I_{OC} \times R_6) = 10 - (1 \times 0.1) = 0.9V$

   $V_H = V_S - (I_{RC} \times R_6) = 10 - (0.5 \times 0.1) = 0.95V$

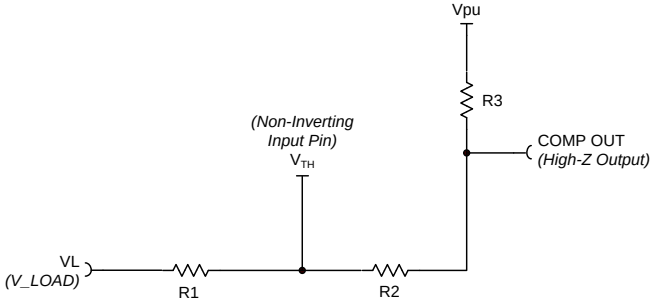


3. With the non-inverting input pin of the comparator labeled as V_{TH} and the comparator output in a logic low state (ground), derive an equation for V_{TH} where V_H represents the load voltage (V_{LOAD}) when the comparator output transitions from low to high. Note that the simplified diagram for deriving the equation shows the comparator output as ground (logic low).



$$V_{TH} = V_H \times \left(\frac{R_2}{R_1 + R_2}\right)$$

4. With the non-inverting input pin of the comparator labeled as V_{TH} and the comparator output in a high-impedance state, derive an equation for V_{TH} where V_L represents the load voltage (V_{LOAD}) when the comparator output transitions from high to low. Applying "superposition" theory to solve for V_{TH} is recommended.



$$V_{TH} = V_L \times \left(\frac{R_2 + R_3}{R_1 + R_2 + R_3}\right) + V_{PU} \times \left(\frac{R_1}{R_1 + R_2 + R_3}\right)$$

5. Eliminate variable V_{TH} by setting the two equations equal to each other and solve for R_1. The result is the following quadratic equation. Solving for R_2 is less desirable since there are more standard values for small resistor values than the larger ones.

   $$0 = (V_{PU}) \times R_1^2 + (V_{PU} \times R_2 + V_L \times (R_3 + R_2) - V_H \times R_2) \times R_1 + (V_L - V_H) \times (R_2^2 + R_2 \times R_3)$$

6. Calculate R_1 after substituting in numeric values for V_{PU}, R_2, V_L, V_H, and R_3. For this design, set V_{PU}=3.3, R_2=2M, V_L=9.9, V_H=9.95, and R_3=1k. Please note that R_3 is significantly smaller than R_2 (R_3<<R_2). Increasing R_3 will cause the comparator logic high output level to increase beyond V_{PU} and should be avoided. For example, increasing R_3 to a value of 100k can cause the logic high output to be 3.6V.

   $0 = (3.3) \times R_1^2 + (6.591M) \times R_1 - (200.1G)$

   the positive root for $R_1 = 29.9k\Omega$

   using standard 1% resistor values, $R_1 = 30.1k\Omega$

7. Calculate V_{TH} using the equation derived in Design Step 3; use the calculated value for R_1. Note that V_{TH} is less than V_L since V_{PU} is less that V_L.

   $V_{TH} = V_H \times \left(\frac{R_2}{R_1 + R_2}\right) = 9.802V$

8. With the inverting terminal labeled as V_{TH}, derive an equation for V_{TH} in terms of R_4, R_5, and V_S.

   $V_{TH} = V_S \times \left(\frac{R_5}{R_4 + R_5}\right)$

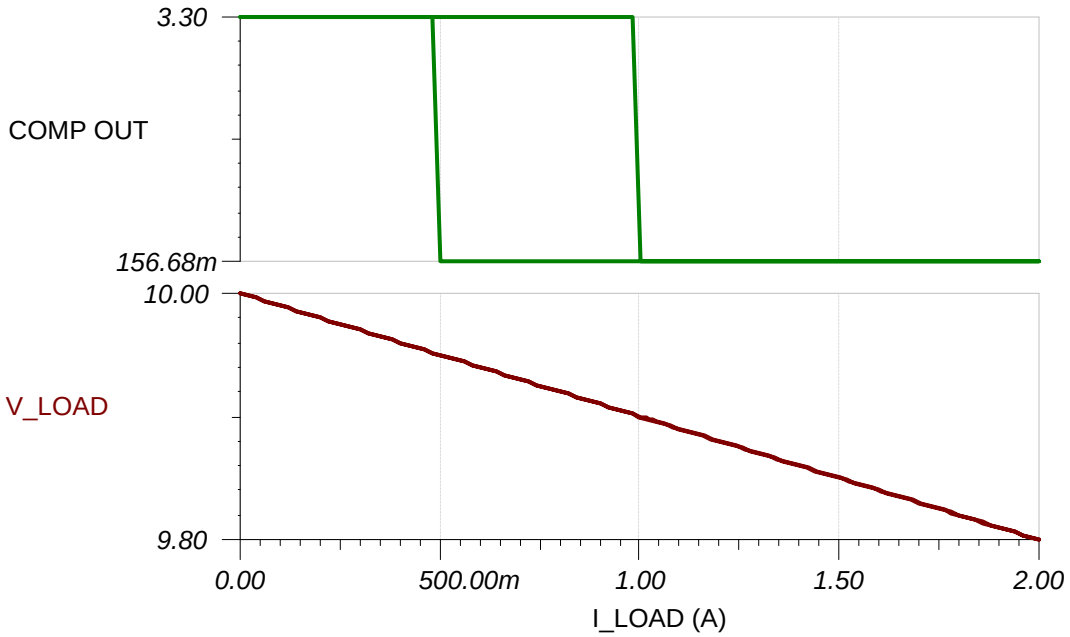
9. Calculate R_4 after substituting in numeric values R_5=1M, V_S=10, and the calculated value for V_{TH}.

   $R_4 = \left(\frac{R_5 \times (V_S - V_{TH})}{V_{TH}}\right) = 20.15k\Omega$

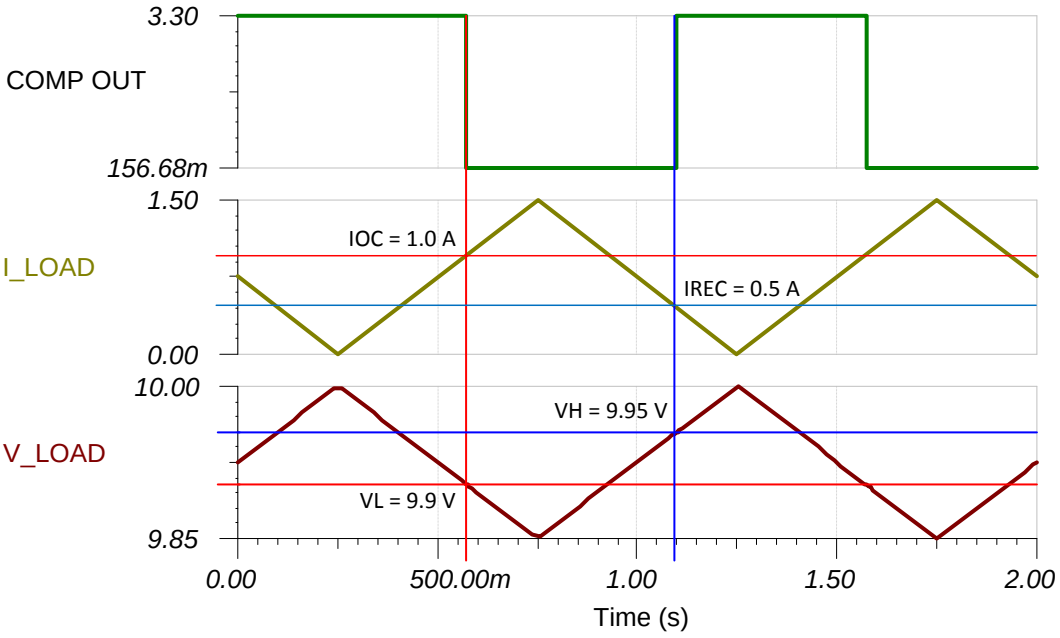
   using standard 1% resistor values, $R_4 = 20.5k\Omega$

Design Simulations

DC Simulation Results



Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See Circuit SPICE Simulation File SLOM456, http://www.ti.com/lit/zip/slom456.

Design Featured Comparator

TLV170x-Q1, TLV170x	
V_S	2.2 V to 36 V
V_{inCM}	Rail-to-rail
V_{OUT}	Open-Drain, Rail-to-rail
V_{OS}	500µV
I_Q	55 µA/channel
$t_{PD(HL)}$	460 ns
#Channels	1, 2, 4
www.ti.com/product/tlv1701-q1	

Design Alternate Comparator

	TLV7021	TLV370x-Q1, TLV340x
V_S	1.6 V to 5.5 V	2.7 V to 16 V
V_{inCM}	Rail-to-rail	Rail-to-rail
V_{OUT}	Open-Drain, Rail-to-rail	Push-Pull, Rail-to-rail
V_{OS}	500 µV	250 µV
I_Q	5 µA	560 µA/Ch
$t_{PD(HL)}$	260 ns	36 µs
#Channels	1	1, 2, 4
	www.ti.com/product/tlv7021	www.ti.com/product/tlv3701-q1

High-speed overcurrent detection circuit

Design Goal

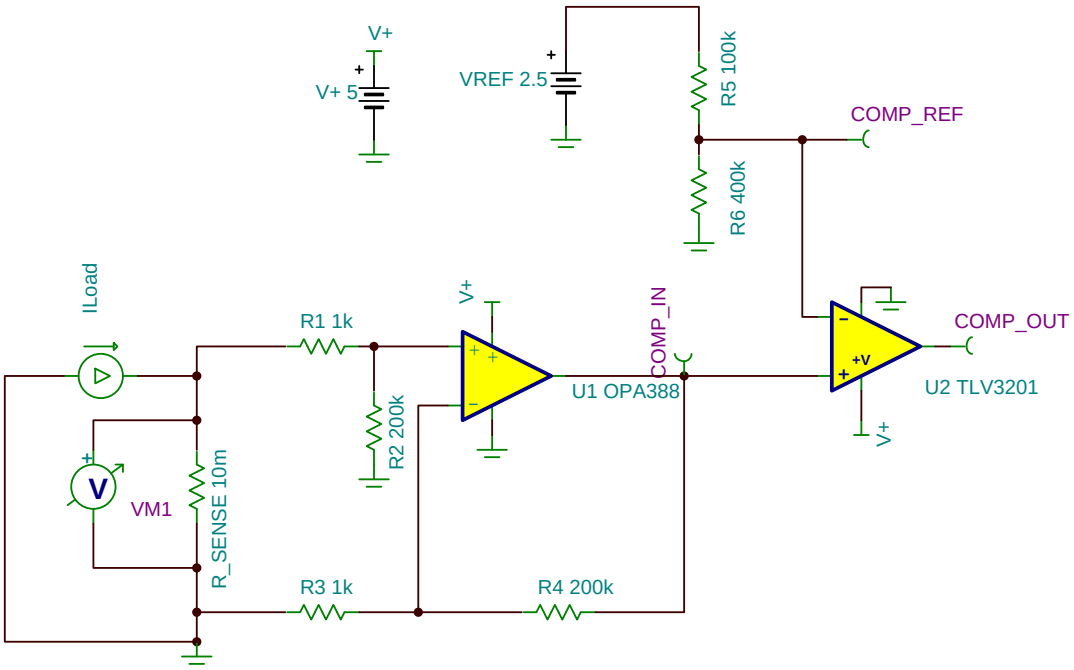
Overcurrent Levels		Supply		Transient Response Time
I_{IN} (min)	I_{IN} (max)	V+	V–	t
0A	1.0A	5V	0V	< 10µs

Design Description

This high-speed, low-side overcurrent detection solution is implemented with a single zero-drift fast-settling amplifier (OPA388) and one high-speed comparator (TLV3201). This circuit is designed for applications that monitor fast current signals and overcurrent events, such as current detection in motors and power supply units.

The OPA388 is selected for its widest bandwidth with ultra-low offset and fast slew rate. The TLV3201 is selected for its fast response due to its small propagation delay of 40ns and rise time of 4.8ns. This allows the comparator to quickly respond and alert the system of an overcurrent event all within the transient response time requirement. The push-pull output stage also allows the comparator to directly interface with the logic levels of the microcontroller. The TLV3201 also has low power consumption with a quiescent current of 40µA.

Typically for low-side current detection, the amplifier across the sense resistor can be used in a noninverting configuration. The application circuit shown, however, uses the OPA388 as a differential amplifier across the sense resistor. This provides a true differential measurement across the shunt resistor and can be beneficial in cases where the supply ground and load ground are not necessarily the same.



Design Notes

1. To minimize errors, choose precision resistors and set $R_1 = R_3$, and $R_2 = R_4$.
2. Select R_{SENSE} to minimize the voltage drop across the resistor at the max current of 1 A.
3. Due to the ultra-low offset of the OPA388 (0.25 µV), the effect of any offset error from the amplifier is minimal on the mV range measurement across R_{SENSE} .
4. Select the amplifier gain so COMP_IN reaches 2 V when the system crosses its critical overcurrent value of 1 A.
5. Traditional bypass capacitors are omitted to simplify the application circuit.

Design Steps

1. Determine the transfer equation where $R_1 = R_3$ and $R_2 = R_4$.

$$COMP\_IN = \left(R_{SENSE} \cdot I_{LOAD}\right) \cdot \left(\frac{R_2}{R_1 + R_2}\right) \cdot \left(1 + \frac{R_4}{R_3}\right)$$

2. Select the SENSE resistor value assuming a maximum voltage drop of 10 mV with a load current of 1 A in order to minimize the voltage drop across the resistor.

$$R_{SENSE} = \frac{V_{SENSE}(max)}{I_{LOAD}(critical)} = \frac{10mV}{1A} = 10m\Omega$$

3. Select the amplifier gain such that COMP_IN reaches 2V when the load current reaches the critical threshold of 1A.

$$Gain = \frac{VREF}{R_{SENSE} \cdot I_{LOAD}(critical)} = \frac{2\ V}{0.01\ V} = \frac{R_2}{R_1 + R_2} \cdot 1 + \frac{R_4}{R_3} = 200$$

Set:

$$R_1 = R_3 = 1k\Omega$$

$$R_2 = R_4 = 200k\Omega$$

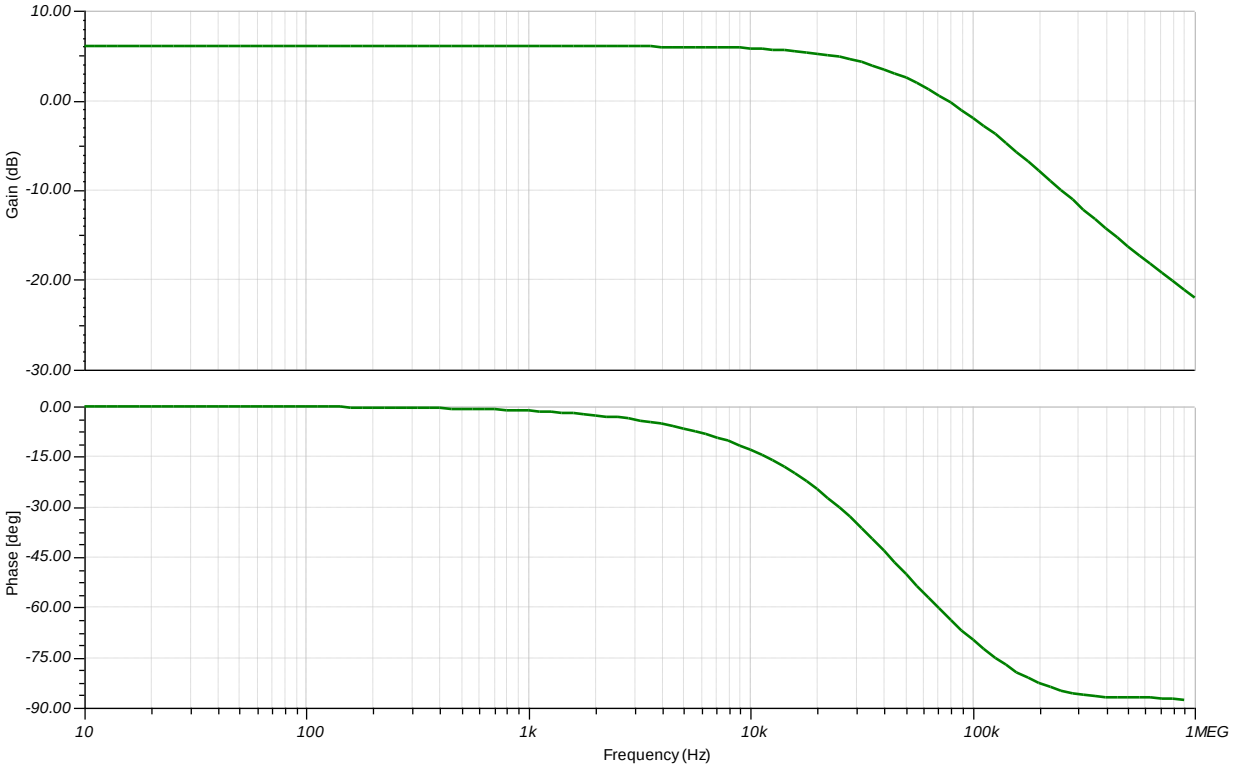
4. Calculate the transimpedance gain of the amplifier in order to verify the following AC simulation results:

$$V_{OUT} = I_{LOAD} \cdot 10m\Omega \cdot 200$$

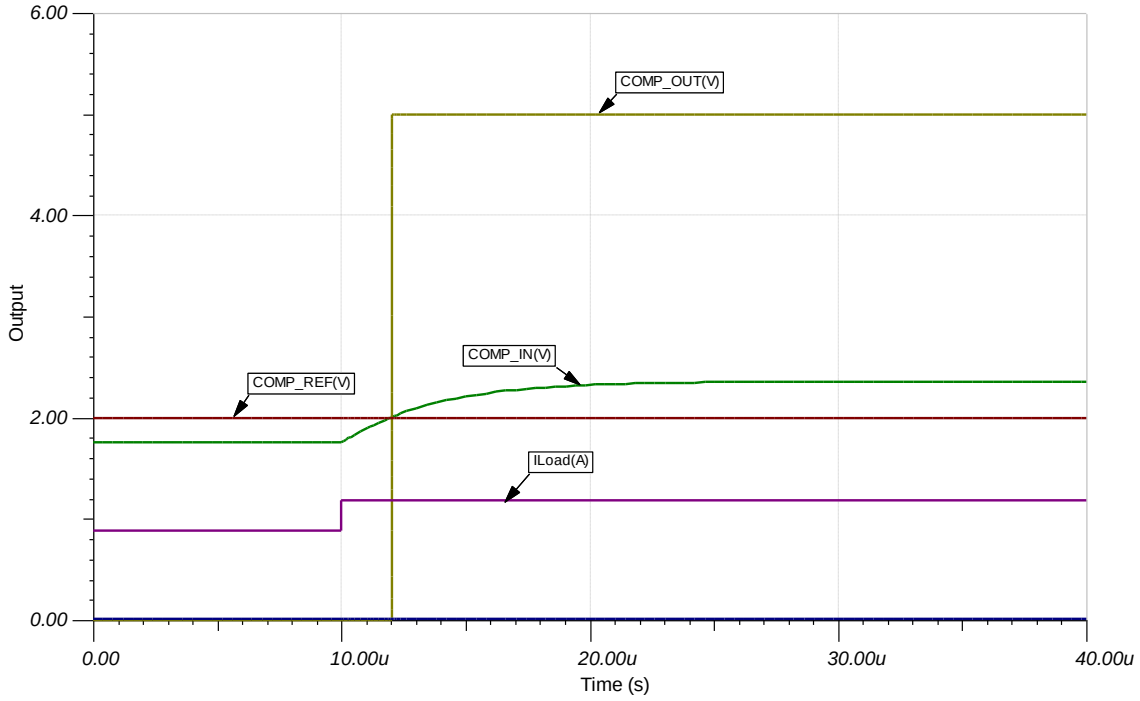
$$\frac{V_{OUT}}{I_{LOAD}} = 10m\Omega \cdot 200 = 2$$

Design Simulations

COMP_IN Transimpedance AC Simulation Results



Transient Response Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the Current sensing using nanopower op amps blog.

References

1. Texas Instruments, Advantages of using nanopower, zero drift amplifiers for battery voltage and current monitoring in portable applications TI tech note
2. Texas Instruments, Current sensing in no-neutral light switches TI tech note
3. Texas Instruments, GPIO Pins power signal chain in personal electronics running on Li-Ion batteries TI tech note

Design Featured Comparator

TLV3201	
V_S	2.7V to 5.5V
t_{PD}	40ns
Input V_{CM}	Rail-to-rail
V_{os}	1mV
I_q	40µA
TLV3201	

Design Alternate Comparator

TLV7021	
V_S	1.6V to 5.5V
t_{PD}	260ns
Input V_{CM}	Rail-to-rail
V_{os}	0.5mV
I_q	5µA
TLV7021	

Design Featured Op Amp

OPA388	
V_S	2.5V to 5.5V
Input V_{CM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	0.25µV
V_{os} Drift	.005µV/°C
I_q	1.7mA/Ch
I_b	30pA
UGBW	10MHz
OPA388	

Design Alternate Op Amp

THS4521	
V_S	2.5V to 5.5V
Input V_{CM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	20 µV
V_{os} Drift	µV/°C
I_q	1mA/Ch
I_b	0.6 µA
UGBW	145MHz
THS4521	

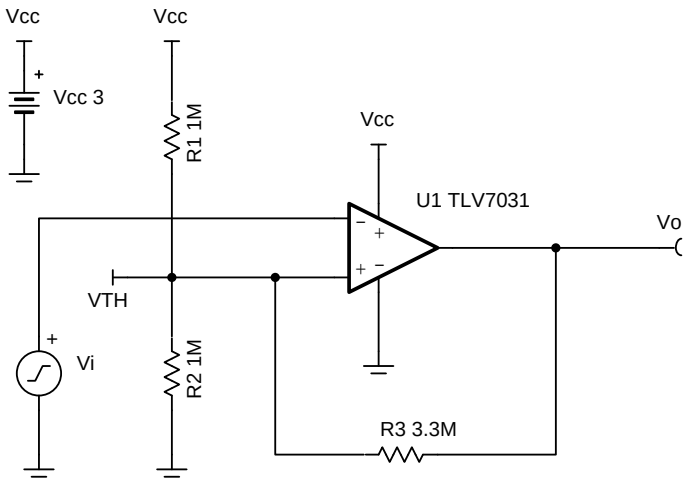
Inverting comparator with hysteresis circuit

Design Goals

Output		Threshold	Hysteresis	Supply	
V_0 = HIGH	V_o = LOW	V_{TH}	V_{HYS}	V_{cc}	V_{ee}
$V_i < V_L$	$V_i > V_H$	1.5V	400mV	3V	0V

Design Description

Comparators are used to differentiate between two different signal levels. When setup in an inverting fashion, the comparator output will be a digital high if the analog input is below a selected threshold. With noise, signal variation, or slow-moving signals at the comparison threshold, undesirable transitions at the output can be observed. Setting upper and lower hysteresis thresholds eliminates these undesirable output transitions. This circuit example will focus on the steps required to design the positive feedback resistor network necessary to obtain the desired hysteresis for an inverting comparator application.



Design Notes

1. The accuracy of the hysteresis threshold voltages are related to the tolerance of the resistors used in the circuit, the selected comparator's input offset voltage specification, and any internal hysteresis already applied to the device.
2. For the TLV7031, V_{OH} is approximately 200mV below V_{cc} and V_{OL} is approximately 250mV above V_{ee}.
3. The TLV7031 has a push-pull output stage, so no pull-up resistor is needed.

Design Steps

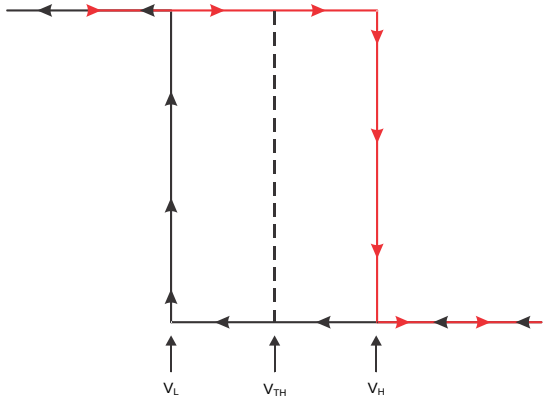
1. Select R_1. This can be a high resistance value due to the very low input bias current caused by the CMOS input of the device.

   $R_1 = 1M\,\Omega$ (Standard Value)

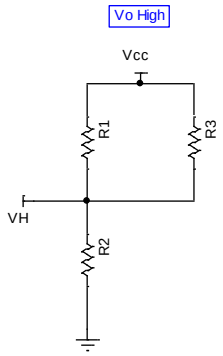
2. Solve for R_2 based on the desired threshold voltage. Set V_{TH} to be 50% of V_{cc} for balanced hysteresis.

   $$R_2 = \frac{R_1 \times V_{TH}}{V_{cc} - V_{TH}} = \frac{1M\Omega \times 1.5V}{3V - 1.5V} = 1M\,\Omega$$

3. Observe the feedback resistor network in the two possible output states: High and Low. Note that the threshold voltage applied to the non-inverting pin by the voltage divider (R_1 and R_2) can be further controlled by using the feedback resistor (R_3). Below is the hysteresis eye diagram.



4. Derive the equation for V_H, which is the threshold voltage when V_o is high. For simplicity, assume V_o switches to V_{cc} when $V_i < V_L$. When this occurs, R_1 and R_3 are in parallel.



5. For push-pull outputs.

   $$V_H = V_{cc} \times \frac{R_2}{(R_1 || R_3) + R_2}$$

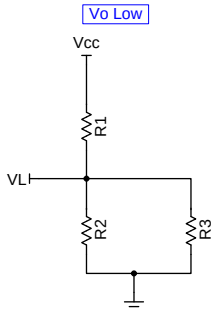
   a. If the comparator in use has an open-drain or open-collector output stage, then the pull-up resistor, R_{pu}, will be in series with R_3. The following equation is true if $V_{pu} = V_{cc}$. Do note that for some applications the pull-up resistor could be ignored in the V_H equation since the eventual feedback resistor value could be significantly larger (ideally 10 times larger) than the pull-up resistor.

   $$V_H = V_{cc} \times \frac{R_2}{\left[R_1 || (R_3 + R_{pu})\right] + R_2}$$

   b. If $V_{pu} \neq V_{cc}$, then use the following equation for V_H.

   $$V_H = \frac{(R_1 \times V_{pu} + (R_3 + R_{pu}) \times V_{cc}) \times R_2}{R_1 \times (R_2 + R_3 + R_{pu}) + R_2 \times (R_3 + R_{pu})}$$

6. Derive the equation for V_L, which is the threshold voltage when V_o is low. For simplicity, assume V_o switches to V_{ee} when $V_i > V_H$. When this occurs, R_2 and R_3 are in parallel.



$$V_L = V_{cc} \times \frac{R_2 || R_3}{R_1 + (R_2 || R_3)}$$

7. Derive the equation for V_{HYS}.

$$V_{HYS} = V_H - V_L = \frac{R_1 \times R_2 \times V_{cc}}{R_1 \times (R_2 + R_3) + (R_2 \times R_3)}$$

8. Solve for R_3.

$$R_3 = \frac{R_1 \times R_2 \times (V_{cc} - V_{HYS})}{(R_1 + R_2) \times V_{HYS}} = \frac{1M\Omega \times 1M\Omega \times (3V - 0.4V)}{(1M\Omega + 1M\Omega) \times 0.4V} = 3.25M\Omega$$

$$R_3 = 3.3M\Omega \quad \text{(Standard Value)}$$

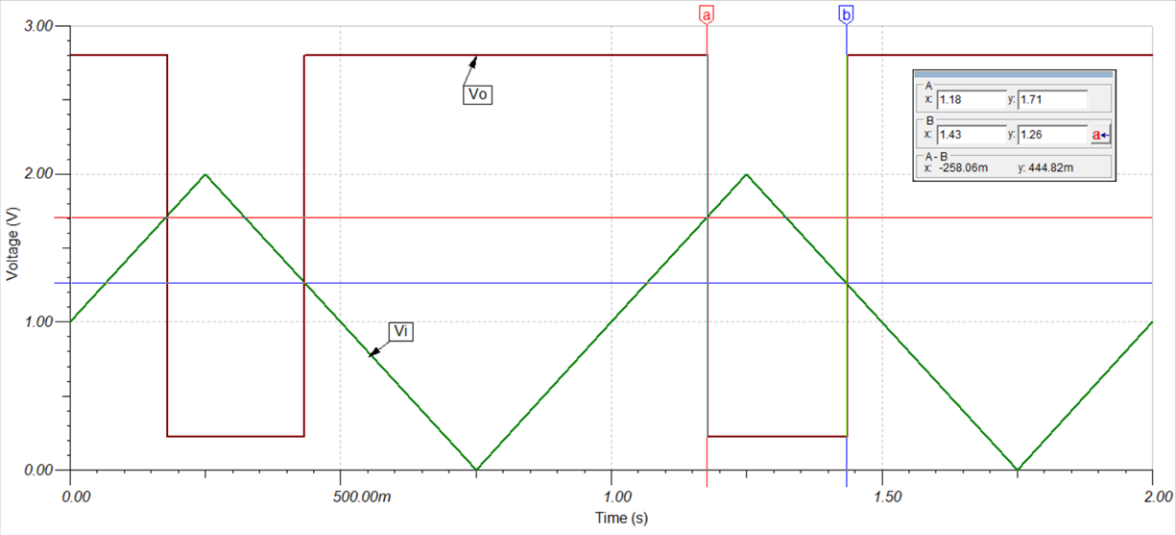
9. Verify V_{HYS}=400mV such that V_H = 1.7V and V_L = 1.3V.

$$V_H = V_{cc} \times \frac{R_2}{(R_1 || R_3) + R_2} = 3V \times \frac{1M\Omega}{(1M\Omega || 3.3M\Omega) + 1M\Omega} = 1.70V$$

$$V_L = V_{cc} \times \frac{R_2 || R_3}{R_1 + (R_2 || R_3)} = 3V \times \frac{(1M\Omega || 3.3M\Omega)}{1M\Omega + (1M\Omega || 3.3M\Omega)} = 1.30V$$

$$V_{HYS} = V_H - V_L = 1.70V - 1.30V = 400mV$$

Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See Comparator with Hysteresis Reference Design TIPD144, www.ti.com/tipd144.

See Circuit SPICE Simulation File SLVMCQ0, http://www.ti.com/lit/zip/slvmcq0.

For more information on many comparator topics including hysteresis, propagation delay and input common mode range please see training.ti.com/ti-precision-labs-op-amps.

Design Featured Comparator

TLV7031	
Output Type	Push-Pull
V_{cc}	1.6V to 6.5V
V_{inCM}	Rail-to-rail
V_{os}	±100µV
V_{HYS}	7mV
I_q	335nA/Ch
t_{pd}	3µs
#Channels	1
	www.ti.com/product/tlv7031

Design Alternate Comparator

TLV1701	
Output Type	Open Collector
V_{cc}	2.2V to 36V
V_{inCM}	Rail-to-rail
V_{HYS}	N/A
V_{os}	±500µV
I_q	55µA/Ch
t_{pd}	560ns
#Channels	1, 2, 4
	www.ti.com/product/tlv1701

Low-power, bidirectional current-sensing circuit

Design Goals

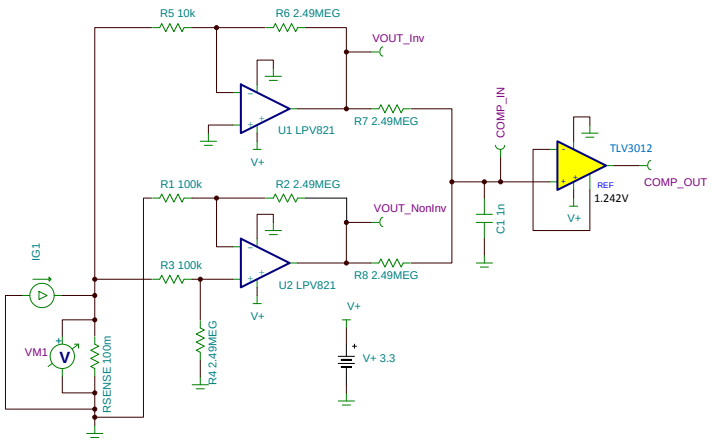
Overcurrent Levels		Supply	
I_{IN} (min)	I_{IN} (max)	V+	V–
–0.1 A	1.0 A	3.3V	0V

Design Description

This low-power, low-side, bidirectional current sensing solution uses two nano-power, zero-drift amplifiers (LPV821) and one micro-power comparator with an integrated, precision reference (TLV3012). This circuit is well-suited for battery powered devices where charging current and system current need to be monitored accurately. The gain of U1 and U2 are set independently.

As shown in the application circuit, the LPV821 amplifiers are connected out of phase across R_{SENSE} to amplify the currents of opposite polarity. Amplifier U2 linearly amplifies the charging (positive) current while amplifier U1 linearly amplifies the system (negative) current. When U2 is monitoring the positive current, U1 drives its output to ground. Similarly, U2 drives its ouput to ground when U1 monitors the negative current. The amplifier outputs are ORed together with resistors R_7 and R_8 while U1 or U2 provide the ground reference creating a single output voltage for the comparator to monitor.

If a regulated supply or reference is already available in the system, the TLV3012 can be replaced by a nano-power comparator such as the TLV7031. Moreover, if the charging current and system current have equal magnitudes, the gains of amplfier U1 and U2 can be set equal to each other. Even with the gains of the amplifiers being equal, ORing the amplifier outputs allows one comparator to detect overcurrent conditions for both charging and system current.



Design Notes

1. To minimize errors, utilize precision resistors and set $R_1 = R_3$, $R_2 = R_4$, and $R_7 = R_8$.
2. Select R_{SENSE} to minimize the voltage drop at max current and to reduce amplifier offset error when monitoring minimum current levels.
3. Select the amplifier gains so COMP_IN reaches 1.242V when the charging and system currents reach their critical levels and avoid operating the amplifiers outside of their linear range.

Design Steps

1. Determine the transfer equation given $R_1 = R_3$, $R_2 = R_4$, and $R_7 = R_8$.

   Inverted Path:

   $$COMP\_IN = -I_{G1} \times R_{SENSE} \times \left(-\frac{R_6}{R_5}\right) \times \left(\frac{R_8}{R_7 + R_8}\right)$$

   Non-Inverted Path:

   $$COMP\_IN = I_{G1} \times R_{SENSE} \times \left(\frac{R_4}{R_3 + R_4}\right) \times \left(\frac{R_1 + R_2}{R_1}\right) \times \left(\frac{R_7}{R_7 + R_8}\right)$$

2. Select the SENSE resistor value assuming a maximum voltage drop (V_{SENSE}) of 100mV when charging at 1A and a minimum system current of 10mA.

   $$R_{SENSE}\ (max) = \frac{V_{SENSE}\ (max)}{I_{G1}\ (max)} = \frac{100\ mV}{1\ A} = 100\ m\Omega$$

   $$\text{with}\ \ I_{G1}(min) = 10mA,\quad V_{SENSE} = 10mA \times 100m\Omega = 1\ mV > > VOS(max) = 10\ \mu V$$

3. Select ORing resistor R_7 and R_8 to generate COMP_IN.

   a. An equal attenuation factor of two is applied to the input of the comparatorwith $R_7 = R_8$. Choose large values to minimize current consumption from the output of the amplifiers.

   b. Special care must be taken when validating the voltage at COMP_IN. Since R_7 and R_8 are large impedance values, the input impedance of an oscilloscope probe or the input to a digital voltmeter can alter the measured voltage. Common probe and voltmeter input impedances are 10MΩ and this will attenuate the signal measured.

   $$\text{with}\ \ R_7 = R_8 = 2.49\ M\Omega,$$
   $$COMP\_IN = (VOUT\_Inv\ \ or\ \ VOUT\_NonInv) / 2$$

4. Select the amplifier gain such that COMP_IN reaches 1.242V when the currents reach the critical threshold.

   $$Gain = \frac{2 \times Comparator\ \ REF}{R_{SENSE} \times |I_{G1}\ (max)|}$$

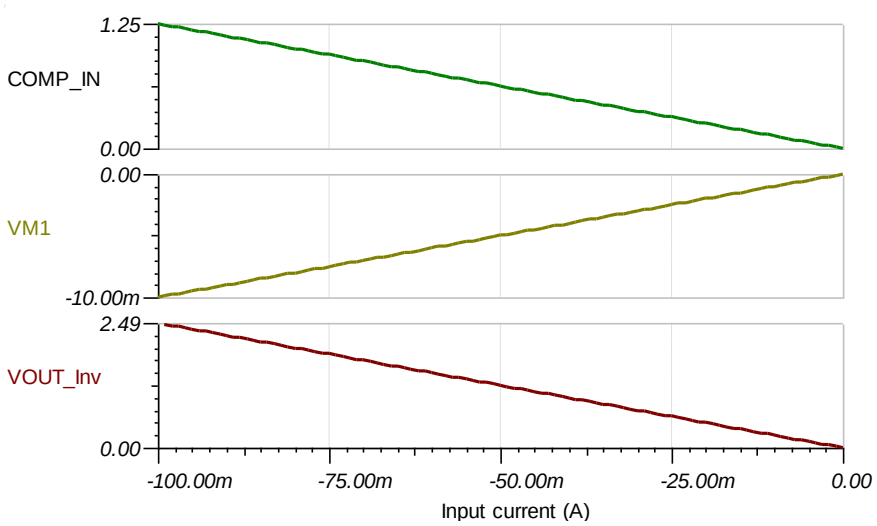
   $$Gain\ \ (Inv) = \frac{2 \times 1.242}{0.1 \times (-0.1)} = \frac{(-R_6)}{R_5} \approx -249\frac{V}{V}$$

   $$Gain\ \ (NonInv) = \frac{2 \times 1.242}{0.1 \times 1.0} = \frac{R_4}{R_3 + R_4} \times \frac{R_1 + R_2}{R_1} \approx 24.9\frac{V}{V}$$

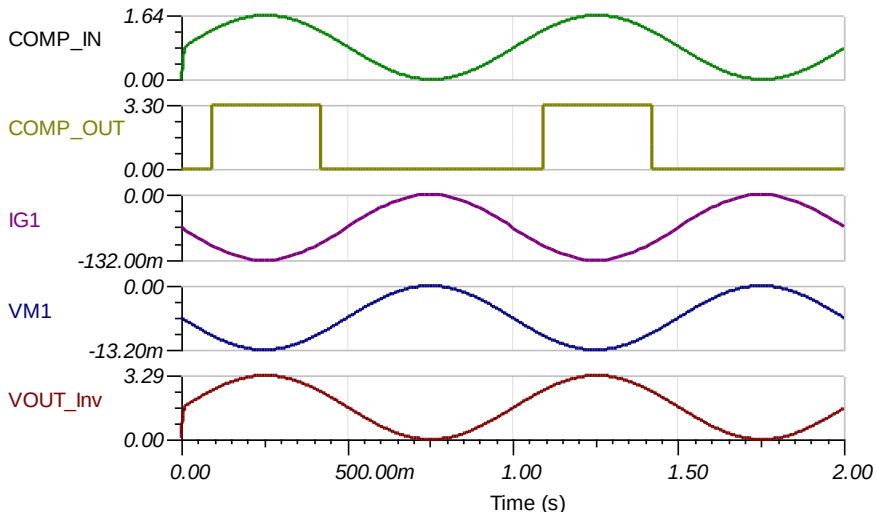
   $R_1 = R_3 = 100\ k\Omega$ (Standard Value)
   $R_5 = 10\ k\Omega$ (Standard Value)
   $R_2 = R_4 = R_6 = 2.49\ M\Omega$ (Standard Value)

Design Simulations

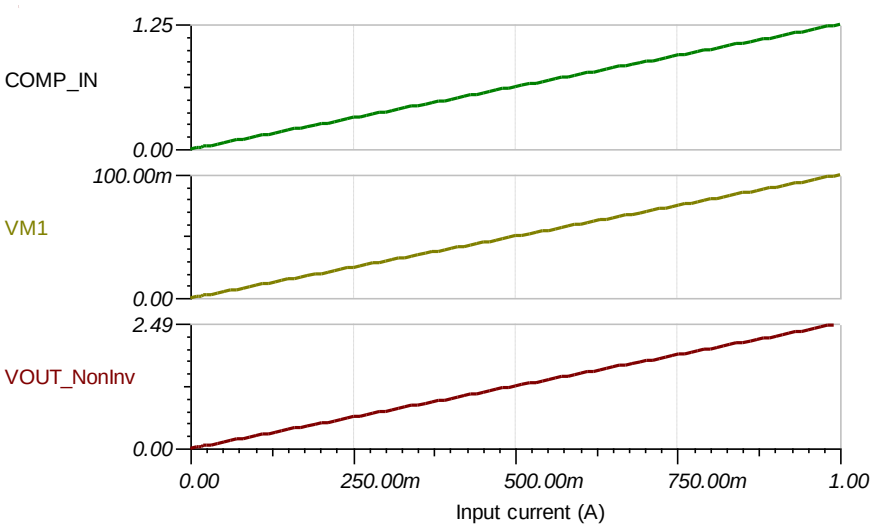
DC Simulation Results (VOUT_Inv)



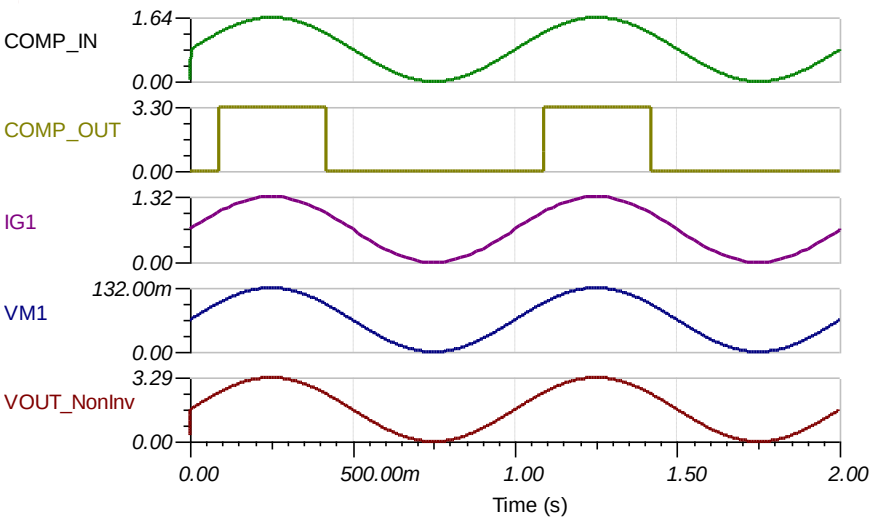
Transient Simulation Results (VOUT_Inv)



DC Simulation Results (VOUT_NonInv)



Transient Simulation Results (VOUT_NonInv)



Tech Note and Blog References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See Advantages of Using Nanopower Zero Drift Amp for Mobile Phone Battery Monitoring.

See Current Sensing in No-Neutral Light Switches.

See GPIO Pins Power Signal Chain in Personal Electronics Running on Li-Ion Batteries.

See Current Sensing Using NanoPower Op Amps Blog.

Design Featured Op Amp

LPV821	
V_S	1.7V to 3.6V
Input V_{CM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	1.5 µV
V_{os} Drift	20 nV/°C
I_q	650 nA/Ch
I_b	7 pA
UGBW	8 kHz
#Channels	1
LPV821	

Design Alternate Op Amp

TLVx333	
V_S	1.8V to 5.5V
Input V_{CM}	Rail-to-rail
V_{out}	Rail-to-rail
V_{os}	2 µV
V_{os} Drift	20 nV/°C
I_q	17 µA/Ch
I_b	70 pA
UGBW	350 kHz
#Channels	1, 2, 4
TLV333	

Revision History

Revision	Date	Change
A	February 2019	Changed title and changed title role to 'Amplifiers'. Added link to circuit cookbook landing page.

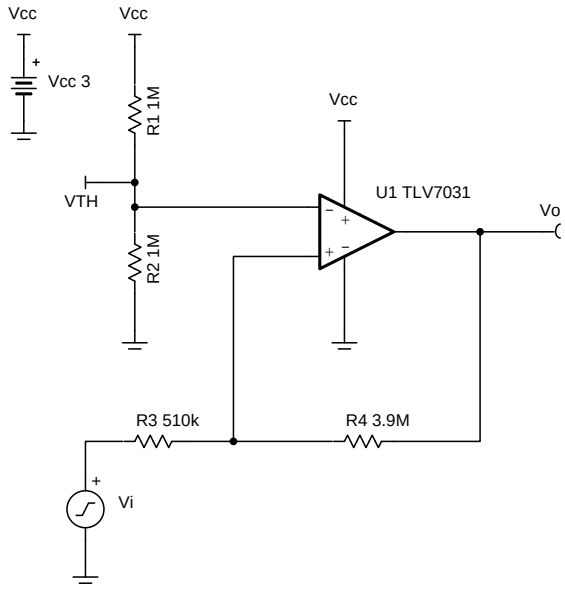
Non-inverting comparator with hysteresis circuit

Design Goals

Output		Hysteresis	Thresholds		Supply	
V_o = HIGH	V_o = LOW	V_{HYS}	V_H	V_L	V_{cc}	V_{ee}
$V_i > V_H$	$V_i < V_L$	400mV	1.7V	1.3V	3V	0V

Design Description

Comparators are used to differentiate between two different signal levels. When setup in a non-inverting fashion, the comparator output will be a digital high if the analog input is above a selected threshold. With noise, signal variation, or a slow-moving signal at the comparison threshold, undesirable transitions at the output can be observed. Setting upper and lower hysteresis thresholds eliminates these undesirable output transitions caused by the noise. This circuit example will focus on the steps required to design the positive feedback resistor network required to obtain the necessary hysteresis for a non-inverting comparator application.



Design Notes

1. Achieving balanced hysteresis will depend on the size of hysteresis and the threshold voltage to V_{cc} ratio.
2. In comparison to the inverting comparator circuit, this example has a lower impedance seen at the inputs.
3. The accuracy of the hysteresis threshold voltages are related to the tolerance of the resistors used in the circuit, the selected comparator's input offset voltage specification, and any internal hysteresis already applied to the device.
4. For the TLV7031, V_{OH} is approximately 200mV below V_{cc} and V_{OL} is approximately 250mV above V_{ee}.
5. The TLV7031 has a push-pull output stage, so no pull-up resistor is needed.

Design Steps

1. Select R_1. This can be a high resistance value due to the very low input bias current caused by the CMOS input of the device.

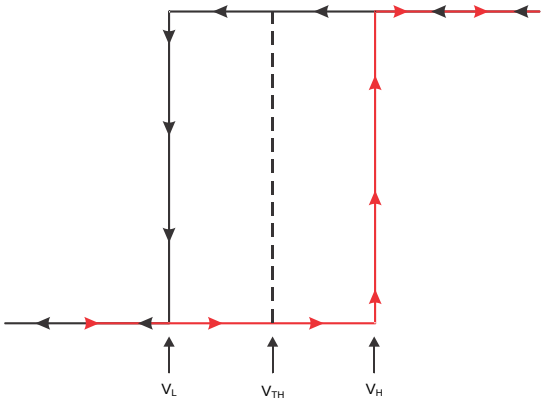
   $R_1 = 1M\,\Omega$ (Standard Value)

2. Solve for R_3. A common practice is to select R_3 to be the impedance seen at the inverting pin to provide input bias current cancellation. Since R_2 is not known, approximate R_3. Here V_{TH} is expected to be 50% of V_{cc}.

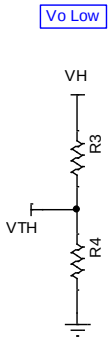
   $R_3 = R_1 || R_2 \cong \frac{1}{2} R_1 = 500k\Omega$

   $R_3 = 510k\Omega$ (Standard Value)

3. Observe the feedback resistor network in the two possible output states: Low and High. Note that the input signal plays a role in determining the hysteresis. The hysteresis eye diagram follows.

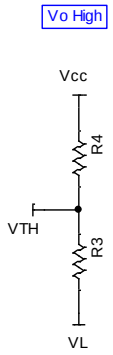


4. Derive the equation for V_H, the voltage that the input signal must rise to for the output to switch high. Set the voltage at the non-inverting pin to be equal to V_{TH}. This ensures the correct hysteresis window will be achieved.



$$V_H = R_3 \times \frac{V_{TH}}{R_4} + V_{TH}$$

5. Derive the equation for V_L, the voltage that the input signal must drop to for the output to switch low. Again, set the voltage at the non-inverting pin to be equal to V_{TH}.



6. For push-pull outputs.

$$V_L = \frac{V_{TH} \times (R_3 + R_4) - V_{cc} \times R_3}{R_4}$$

7. If the comparator in use has an open-drain or open-collector output stage, then the pull-up resistor, R_{pu}, will be in series with R_3 and R_4. The following equation is true if $V_{pu} = V_{cc}$. Do note that for some applications, the pull-up resistor could be ignored in the V_L equation since the eventual feedback resistor value could be significantly larger (ideally 10 times larger) than the pull-up resistor.

$$V_L = \frac{V_{TH} \times (R_{pu} + R_4) - (V_{cc} - V_{TH}) \times R_3}{R_4 + R_{pu}}$$

8. If $V_{pu} \neq V_{cc}$, then use the following equation for V_L.

$$V_L = \frac{V_{TH} \times (R_{pu} + R_4) - (V_{pu} - V_{TH}) \times R_3}{R_4 + R_{pu}}$$

9. Derive the equation for V_{HYS}.

$$V_{HYS} = V_H - V_L = V_{cc} \times \frac{R_3}{R_4}$$

10. Solve for R_4.

$$R_4 = \frac{V_{cc}}{V_{HYS}} \times R_3 = \frac{3V}{0.4V} \times 510k\Omega = 3.83M\Omega$$

$$R_4 = 3.9M\Omega \quad \text{(Standard Value)}$$

11. Use the V_H equation found in step 4 to now solve for V_{TH}.

$$V_{TH} = \frac{R_4 \times V_H}{R_3 + R_4} = \frac{3.9M\Omega \times 1.7V}{510k\Omega + 3.9M\Omega} = 1.50V$$

12. Verify the V_{TH} value with the V_L equation found in step 6.

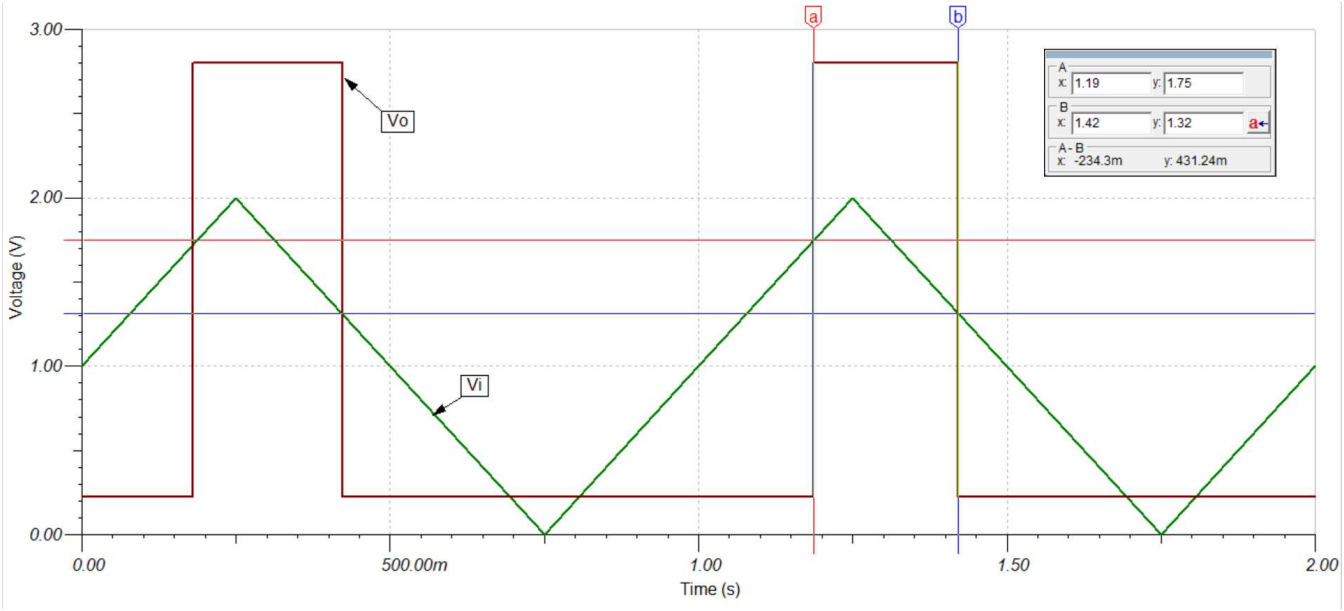
$$V_{TH} = \frac{R_4 \times V_H}{R_3 + R_4} = \frac{3.9M\Omega \times 1.7V}{510k\Omega + 3.9M\Omega} = 1.50V$$

13. Solve for R_2 based on the calculated threshold voltage, V_{TH}.

$$R_2 = \frac{R_1 \times V_{TH}}{V_{cc} - V_{TH}} = \frac{1M\Omega \times 1.5V}{3V - 1.5V} = 1M\,\Omega$$

Design Simulations

Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See Circuit SPICE Simulation File SLVMCR2.

For more information on many comparator topics including hysteresis, propagation delay and input common mode range please see training.ti.com/ti-precision-labs-op-amps.

Design Featured Comparator

TLV7031	
Output Type	Push-Pull
V_{cc}	1.6V to 6.5V
V_{inCM}	Rail-to-rail
V_{os}	±100µV
V_{HYS}	7mV
I_q	335nA/Ch
t_{pd}	3µs
#Channels	1
	www.ti.com/product/tlv7031

Design Alternate Comparator

TLV1701	
Output Type	Open Collector
V_{cc}	2.2V to 36V
V_{inCM}	Rail-to-rail
V_{HYS}	N/A
V_{os}	±500µV
I_q	55µA/Ch
t_{pd}	560ns
#Channels	1, 2, 4
	www.ti.com/product/tlv1701

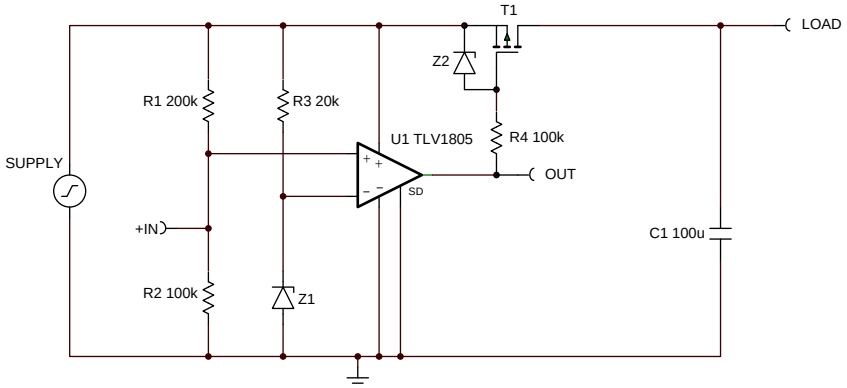
Overvoltage protection with comparator circuit

Design Goals

Supply	Load	Comparator Output Status (OUT)	
Operating Voltage Range	MAX Operating Voltage (V_{OVER})	SUPPLY < V_{OVER}	SUPPLY ≥ V_{OVER}
12V to 36V	30V	V_{OL} < 0.4V	V_{OH} = SUPPLY

Design Description

This overvoltage protection circuit uses a high-voltage comparator with a push-pull output stage to control a P-Channel MOSFET that connects the SUPPLY to the LOAD. When the SUPPLY voltage exceeds the overvoltage threshold (V_{OVER}), the output of the comparator goes HIGH and disconnects the LOAD from the SUPPLY by opening the P-Channel MOSFET. Likewise, when the SUPPLY voltage is below V_{OVER}, the output of the comparator is LOW and connects the LOAD to the SUPPLY.



Design Notes

1. Select a high-voltage comparator with a push-pull output stage.
2. Select a reference voltage that is below the lowest operating voltage range for the SUPPLY.
3. Calculate values for the resistor divider so the critical overvoltage level occurs when the input to the comparator (+IN) reaches the comparator's reference voltage.
4. Limit the source-gate voltage of the P-Channel MOSFET so that it remains below the device's maximum allowable value.

Design Steps

1. Select a high-voltage comparator with a push-pull output stage that can operate at the highest possible SUPPLY voltage. In this application, the highest SUPPLY voltage is 36V.
2. Determine an appropriate reference level for the overvoltage detection circuit. Since the lowest operating voltage for the SUPPLY is 12V, a 10V zener diode (Z_1) is selected for the reference (V_{REF}).
3. Calculate value of R3 by considering the minimum bias current to keep the Z_1 regulating at 10V. A minimum bias current of 100uA is used along with the minimum SUPPLY voltage of 12V.

   $$R_3 = \frac{SUPPLY\ (min) - V_{ZENER}}{I_{BIAS}\ (min)} = \frac{12V - 10V}{100\mu A} = 20\ \ k\Omega$$

4. Calculate the resistor divider ratio needed so the input to the comparator (+IN) crosses the reference voltage (10V) when the SUPPLY rises to the target overvoltage level (V_{OVER}) of 30V.

   $$V_{REF} = V_{OVER} \times \left(\frac{R_2}{R_1 + R_2}\right)$$

   $$\left(\frac{R_2}{R_1 + R_2}\right) = \frac{V_{REF}}{V_{OVER}} = \frac{10V}{30V} = 0.333$$

5. Select values for R_1 and R_2 that yield the resistor divider ratio of 0.333V by using the following equation or using the online "Voltage Divider Calculator" at http://www.ti.com/download/kbase/volt/volt_div3.htm.

   If using the following equation, choose a value for R_2 in the 100k-ohm range and calculate for R_1. In this example, a value of 100k was chosen for R_2.

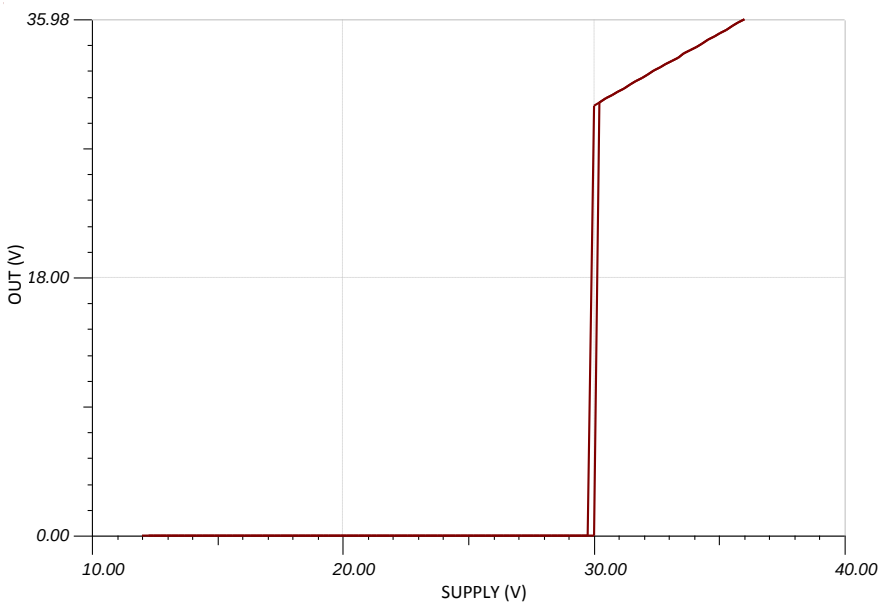
   $$R_1 = R_2\left(\frac{V_{OVER}}{V_{REF}} - 1\right) = 100\ \ k\Omega\ \ \left(\frac{30V}{10V} - 1\right) = 200\ \ k\Omega$$

6. Note that the TLV1805 which is used in application circuit has 15mV of hysteresis. This means that the actualy switching threshold will be 7.5mV higher than the switching threshold (VREF) when the SUPPLY is rising and 7.5mV lower when the SUPPLY is falling. The result of the hysteresis is most easily seen in the DC Simulation curve. Since SUPPLY is resistor divided down by a factor of 3, the net impact to the SUPPLY switching threshold is 3 times this amount.
7. Verify that the current through the resistor divider is at least 100 times higher than the input bias current of the comparator. The resistors can have high values to minimize power consumption in the circuit without adding significant error to the resistor divider.
8. Select a zener diode (Z_2) to limit the source-gate voltage (V_{SG}) of the P-Channel MOSFET so that it remains below the device's maximum allowable value. It is common for P-Channel, power MOSFETs to have a V_{SG} max value of 20V, so a 16V zener is placed from source to gate.
9. Calculate a value for the current limiting resistor (R_4). When SUPPLY rises above 16V and Z_2 begins to conduct, R_4 limits the amount of current that the comparator output will sink when its output is LOW. With a nominal SUPPLY voltage of 24V, the sink current is limited to 80µA.

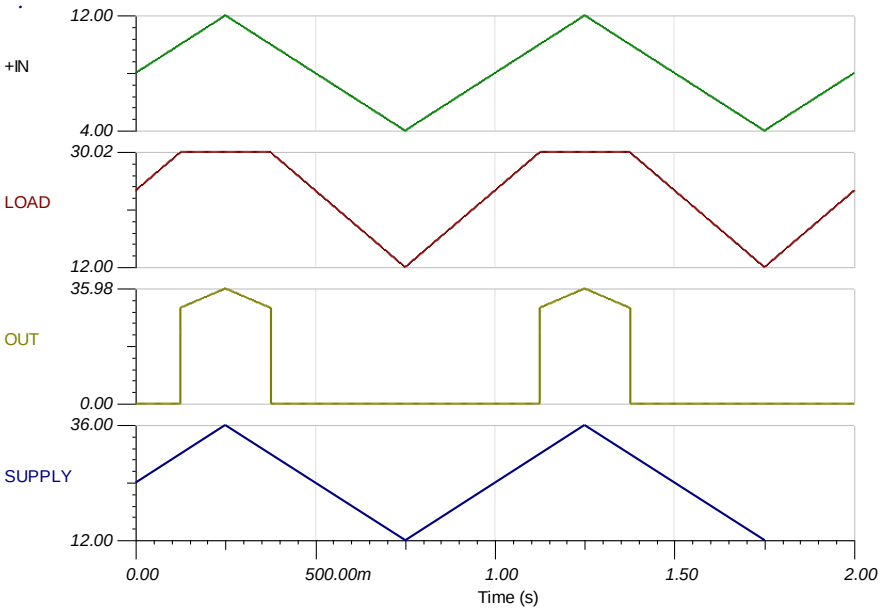
   $$I_{SINK} = \left(\frac{SUPPLY - V_{Z2}}{R_4}\right) = \left(\frac{24V - 16V}{100\ k\Omega}\right) = 80\ \ \mu A$$

Design Simulations

DC Simulation Results



Transient Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SNOAA20
3. TI Precision Labs

Design Featured Comparator

TLV1805-Q1 / TLV1805	
V_S	3.3 V to 40 V
V_{inCM}	Rail-to-rail
V_{OUT}	Push-Pull
V_{OS}	500 µV
Hysteresis	15 mV
I_Q	135 µA
$t_{PD(HL)}$	250 ns
www.ti.com/product/tlv1805	

Design Alternate Comparator

	TLV3701 / TLV370x-Q1	TLC3702 / TLC3702-Q1
V_S	2.5 V to 16 V	4V to 16 V
V_{inCM}	Rail-to-rail	-1 V from VDD
V_{OUT}	Push-Pull	Push-Pull
V_{OS}	250 µV	1.2 mV
Hysteresis	n/a	n/a
I_Q	0.56 µA	9.5 µA/Ch
$t_{PD(HL)}$	36 µs	0.65 µs
	www.ti.com/product/tlv3701	www.ti.com/product/tlc3702

Window comparator with integrated reference circuit

Design Goals

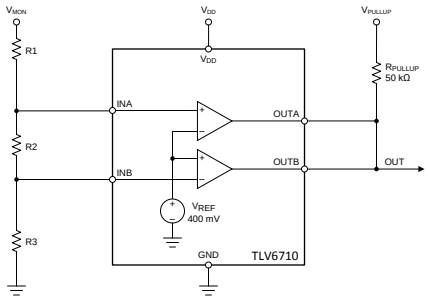
Input		Output		Supply	
$V_{MON\ Min}$	$V_{MON\ Max}$	$V_{OUT\ Min}$	$V_{OUT\ Max}$	V_{DD}	V_{REF}
0V	6V	0V	3.3V	3.3V	400mV

Lower Threshold (V_L)	Upper Threshold (V_H)	Divider Load Current (I_{MAX}) at V_H
3.2V	4.1V	10uA

Design Description

This circuit utilizes the TLV6710, which contains two comparators and a precision internal reference of 400mV. The monitored voltage (V_{MON}) is divided down by R_1, R_2, and R_3. The voltage across R_2 and R_3 is compared to the 400mV internal reference voltage (V_{REF}). If the input signal (V_{MON}) is within the window, the output is high. If the signal level is outside of the window, the output is low.

The TLV6710 will be utilized for this example, which conveniently contains two comparators and a common precision internal reference trimmed to a 400mV threshold. Two discrete comparators and an external reference may also be used.



Design Notes

1. Make sure the comparator input voltage range is not violated at the highest expected V_{MON} voltage.
2. If the outputs are to be combined together (ORed), open collector or open drain output devices must be used.
3. It is also recommended to repeat the following calculations using the minimum and maximum resistor tolerance values and comparator positive and negative offset voltages.
4. The TLV6710 has built-in asymmetrical hysteresis, resulting in the rising edge V_L and falling edge V_H being slightly shifted. Comparators without hysteresis will meet the calculated thresholds.

Design Steps

The resistor divider will be calculated in separate V_H and V_L segments to create 400mV at the appropriate comparator input at the desired threshold voltage.

1. The total divider resistance R_{TOTAL} is calculated from the upper threshold voltage and divider current:
$$R_{TOTAL} = R_1 + R_2 + R_3 = \frac{V_H}{I_{MAX}} = \frac{4.1V}{10\mu A} = 410k\Omega$$
2. The upper threshold voltage is set by the "bottom" divider resistor R_3 going into the INB pin. From the reference voltage and the divider current, the value of R_3 is calculated from:
$$R_3 = \frac{V_{REF}}{I_{MAX}} = \frac{400mV}{10\mu A} = 40k\Omega$$
3. The "middle" resistor R_2 is found by looking at R_2 and R_1 as one resistor, and calculating the value for that total resistance for V_{REF} at V_L, then subtracting out the known R_3:
$$R_2 = ((\frac{R_{TOTAL}}{V_L} \times V_{REF}) - R_3) = ((\frac{410k\Omega}{3.2V} \times 400mV) - 40k\Omega) = 11.25k\Omega$$
4. R_1 is found by taking the total resistance and subtracting the sum of R_2 and R_3:
$$R_1 = R_{TOTAL} - (R_2 + R_3) = 410k\Omega - (11.25k\Omega + 40k\Omega) = 358.75k\Omega$$

   Because these are calculated ideal resistor values, the next closest 0.1% standard resistor values will be used. The following table summarizes the changes due to the resistor value changes and the resulting trip point voltage change.

Nearest 0.1% Resistor Values

Resistor	Calculated Ideal Value	Nearest Standard 0.1% (E192) Value
R_1	358.750 kΩ	361 kΩ
R_2	11.25 kΩ	11.3 kΩ
R_3	40 kΩ	40.2 kΩ

Because the values of the divider string resistors were changed, the resulting new threshold voltages must be calculated. The thresholds are found by multiplying the divider ratio by the reference voltage:

$$V_H = (\frac{R1+R2+R3}{R3}) \times V_{REF} = (\frac{361k\Omega+11.3k\Omega+40.2k\Omega}{40.2k\Omega}) \times 0.4V = 10.26119 \times 0.4V = 4.1045 \quad V$$

$$V_L = (\frac{R1+R2+R3}{R2+R3}) \times V_{REF} = (\frac{361k\Omega+11.3k\Omega+40.2k\Omega}{11.3k\Omega+40.2k\Omega}) \times 0.4V = 8.0097 \times 0.4V = 3.2039 \quad V$$

Ideal and Standard Resistor Thresholds

Threshold	Using Ideal Resistors	Using Standard Resistors	Percent Change
V_H	4.1V	4.1045V	+0.109%
V_L	3.2V	3.2039V	+0.121%

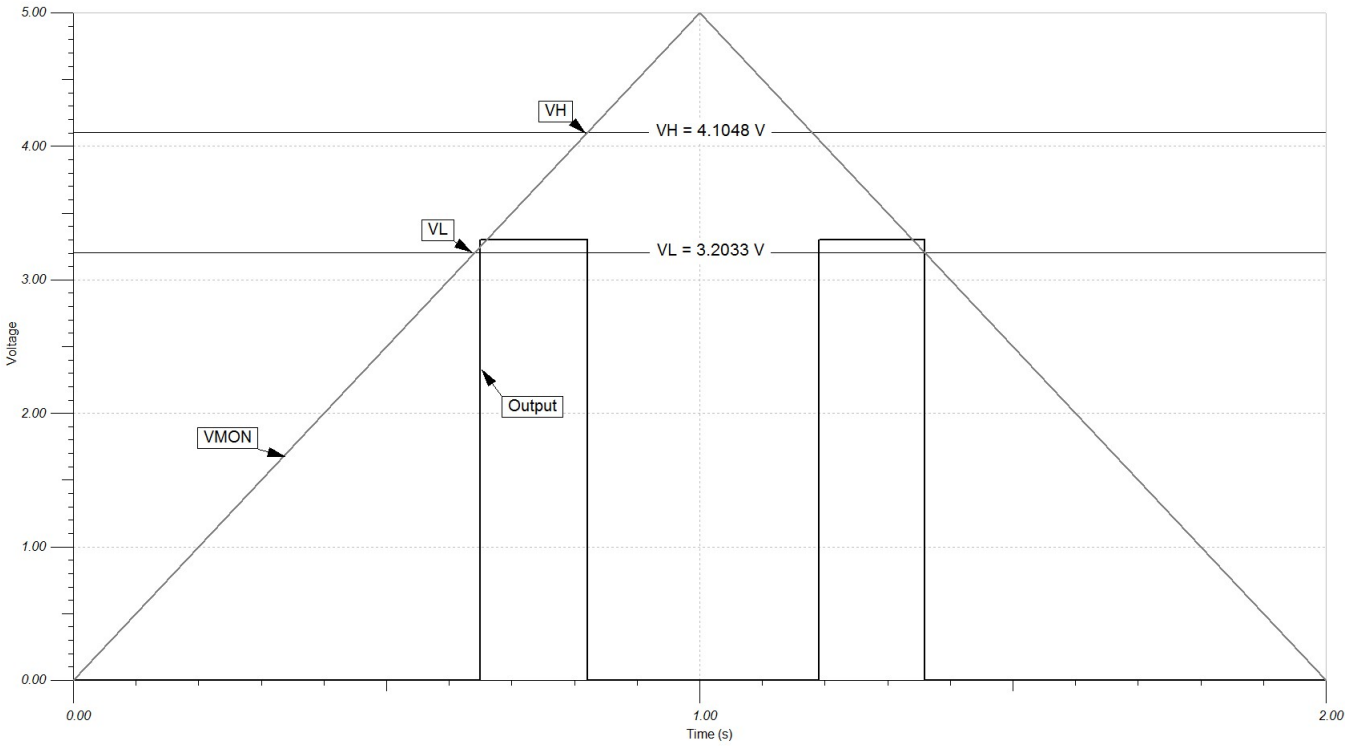
To ensure that the maximum 6V V_{MON} voltage does not violate the TLV6710 1.7V maximum input voltage rating, the V_{MON_MAX} and the V_L division ratio found in step 4 above are used to calculate the maximum voltage at the TLV6710 input:

$$V_{INPUT\_MAX} = \frac{V_{MON\_MAX}}{V_{L\_RATIO}} = \frac{6\ V}{8.0097} = 749.1 \quad mV$$

The value 749mV is less than 1.7V, so the input voltage is well below the input maximum. If using discrete comparators, make sure the votlage is within the specified input common mode range (V_{ICR}) of the device used.

Design Simulations

Transient Simulation Results



Note: The Rising edge V_L and falling edge V_H thresholds are slightly shifted due to the built-in asymmetrical hysteresis of the TLV6710. Comparators without hysteresis will meet the calculated thresholds.

Design References

For more information on many comparator topics including input votlage range, output types and propagation delay, please visit TI Precision Labs - Comparator Applications.

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See TINA-TI™ TLV6710 Reference Design circuit simulation file, Literature Number SNVMB09.

Design Featured Comparator

TLV6710	
V_{ss}	2V to 36 V
V_{inCM}	0V to 1.7V
V_{out}	0V to 25V
Vref	400 mV ±0.25%
I_q	11 µA
I_b	1 nA
Prop Delay	10 µs
#Channels	2
www.ti.com/product/tlv6710	

Design Alternate Comparator

TLV6700	
V_{ss}	1.8V to 18 V
V_{inCM}	0V to 6.5V
V_{out}	0V to 18V
Vref	400 mV ±0.5%
I_q	5.5 µA
I_b	1 nA
Prop Delay	29 µs
#Channels	2
www.ti.com/product/tlv6700	

Design Alternate Comparator

TLV1702	
V_{ss}	2.7 to 36 V
V_{inCM}	Rail to Rail
V_{out}	Open Drain to 36 V
V_{os}	±3.5 mV
I_q	75 µA
I_b	15 nA
Prop Delay	0.4 µs
#Channels	2
www.ti.com/product/tlv1702	

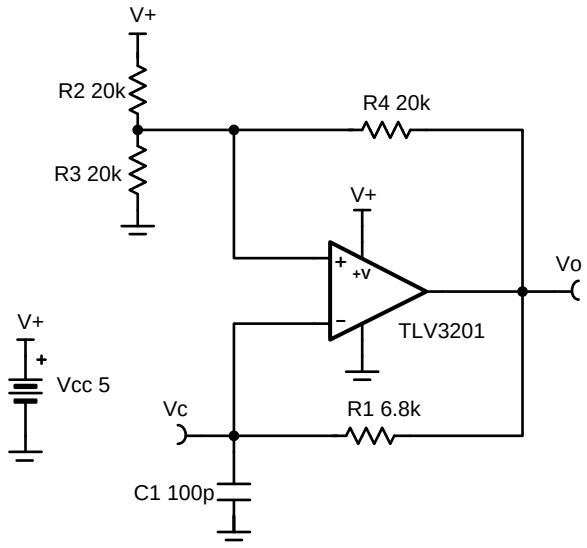
Relaxation oscillator circuit

Design Goals

Supply		Oscillator Frequency
V_{cc}	V_{ee}	f
5V	0V	1 MHz

Design Description

The oscillator circuit generates a square wave at a selected frequency. This is done by charging and discharging the capacitor, C_1 through the resistor, R_1. The oscillation frequency is determined by the RC time constant of R_1 and C_1, and the threshold levels set by the resistor network of R_2, R_3, and R_4.The maximum frequency of the oscillator is limited by the toggle rate of the comparator and the capacitance load at the output. This oscillator circuit is commonly used as a time reference or a supervisor clock source.



Design Notes

1. Comparator toggle rate and output capacitance are critical considerations when designing a high-speed oscillator.
2. Select C_1 to be large enough to minimize the errors caused by stray capacitance.
3. If using a ceramic capacitor, select a COG or NPO type for best stability over temperature.
4. Select lower value resistors for the R_2, R_3, R_4 resistor network to minimize the effects of stray capacitance.
5. R_2, R_3, and R_4 can be adjusted in order to create a duty cycle other than 50%.

Design Steps

1. When $R_2 = R_3 = R_4$, the resistor network sets the oscillator trip points of the non-inverting input at one-third and two-thirds of the supply.
2. When the output is high, the upper trip point will be set at two-thirds of the supply to bring the output back low.

   $$V_o = V_s\left(\frac{R_3}{(R_2 \| R_4)+R_3}\right) = \frac{2}{3}V_s = 3.33V$$

3. When the output is low, the lower trip point will be set at one-third of the supply in order to bring the output back high.

   $$V_o = V_s\left(\frac{R_3 \| R_4}{(R_3 \| R_4)+R_2}\right) = \frac{1}{3}V_s = 1.67V$$

4. The timing of the oscillation is controlled by the charging and discharging rate of the capacitor C_1 through the resistor R_1. This capacitor sets the voltage of the inverting input of the comparator. Calculate the time to discharge the capacitor.

   $$V_c = V_i e^{-\frac{t}{R_1 C_1}}$$

   $$\frac{1.67}{3.33} = e^{-\frac{t}{R_1 C_1}}$$

   $$t = 0.69 R_1 C_1$$

5. Calculate the time to charge the capacitor.

   $$V_i = V_c\left(1 - e^{-\frac{t}{RC}}\right)$$

   $$1.67 = 3.33\left(1 - e^{-\frac{t}{RC}}\right)$$

   $$\frac{1.67}{3.33} = e^{-\frac{t}{RC}}$$

   $$t = 0.69 R_1 C_1$$

6. The time for the capacitor to charge or discharge is given by $0.69R_1C_1$. With a target oscillator frequency of 1MHz, the time to charge or discharge should be 500ns.

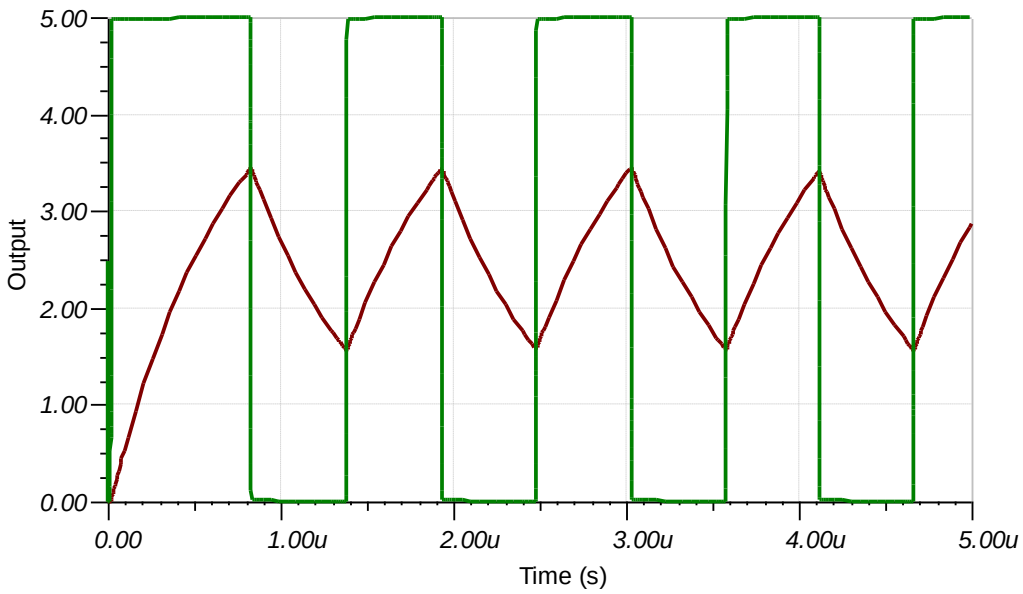
   $$0.69 R_1 C_1 = 500ns$$

   $$R_1 C_1 = 724ns$$

7. Select C_1 as 100 pF and R_1 as 6.8kΩ (the closest real world value).

Design Simulations

Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit spice simulation file, SBOMAO3.

For more information on many comparator topics including hysteresis, propagation delay and input common mode range please see, TI Precision Labs.

Design Featured Comparator

TLV3201	
V_{ss}	2.7 to 5.5V
V_{inCM}	Rail-to-rail
t_{pd}	40ns
V_{os}	1mV
V_{HYS}	1.2mV
I_q	40µA
Output Type	Push-Pull
#Channels	1
www.ti.com/product/tlv3201	

Design Alternate Comparator

TLV7011	
V_{ss}	1.6 to 5.5V
V_{inCM}	Rail-to-rail
t_{pd}	260ns
V_{os}	0.5V
V_{HYS}	4mV
I_q	5µA
Output Type	Push-Pull
#Channels	1
www.ti.com/product/tlv7011	

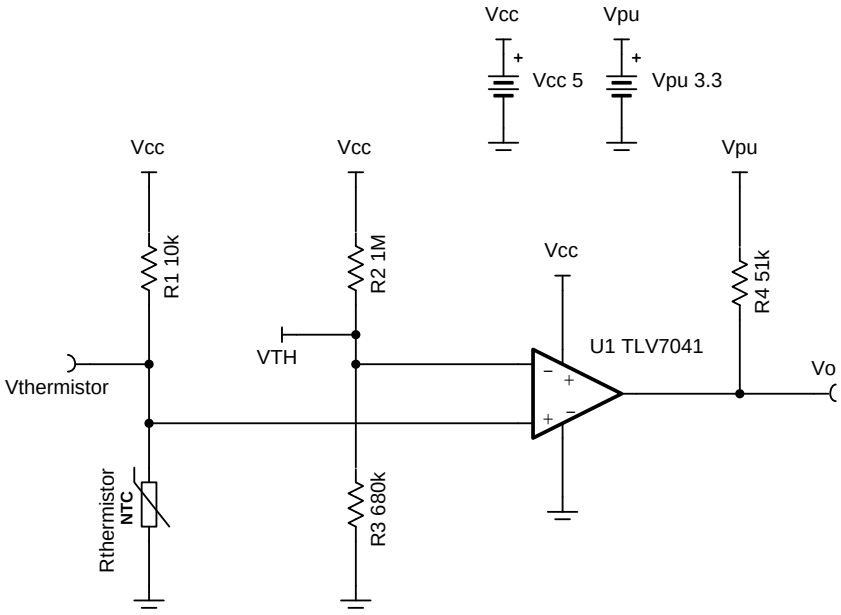
Thermal switch circuit

Design Goals

Temperature Switching Point	Output		Supply		
T_{sp}	V_o = HIGH	V_o = LOW	V_{cc}	V_{ee}	V_{pu}
100 °C	$T_A < T_{sp}$	$T_A > T_{sp}$	5V	0V	3.3V

Design Description

This thermal switch solution will signal low (to a GPIO pin) when a certain temperature is exceeded thus alerting when conditions are no longer optimal or device-safe. This circuit incorporates an NTC thermistor with a comparator configured in a non-inverting fashion.



Design Notes

1. The resistance of an NTC thermistor drops as temperature increases.
2. The TLV7041 has an open drain output, so a pull-up resistor is required.
3. Configurations where the thermistor is placed near the high side of the divider can be done; however, the comparator will have to be used in an inverting fashion to still have the output switch low.
4. To exercise good practice, a positive feedback resistor should be placed to add external hysteresis (for simplicity, it is not done in this example).

Design Steps

1. Select an NTC thermistor, preferably one with a high nominal resistance, R_0, (resistance value when ambient temperature, T_A, is 25 °C) since the TLV7041 has a very low input bias current. This will help lower power consumption, thus reducing the likelihood of reading a slightly higher temperature due to thermal dissipation in the thermistor. The thermistor chosen has its R_0 and its material constant, β, listed below.

   $R_0 = 100k\Omega$

   $\beta = 3977K$

2. Select R_1. For high temperature switching points, R_1 should be 10 times smaller than the nominal resistance of the thermistor. This causes a larger voltage difference per temperature change around the temperature switching point, which helps guarantee the output will switch at the desired temperature value.

   $R_1 = \frac{R_0}{10}$

   $R_1 = \frac{100k\Omega}{10} = 10k\Omega$ (Standard Value)

3. Select R_2. Again, this can be a high resistance value.

   $R_2 = 1M\Omega$ (Standard Value)

4. Solve for the resistance of the thermistor, $R_{thermistor}$, at the desired temperature switching point. Using the β formula is an effective approximation for thermistor resistance across the temperature range of -20 °C to 120 °C. Alternatively, the Steinhart-Hart equation can be used, but several device-specific constants must be provided by the thermistor vendor. Note that temperature values are in Kelvin. Here T_0 = 25 °C = 298.15K.

   $R_{thermistor}(T_{sp}) = R_0 \times e^{\beta \times \left(\frac{1}{T_{sp}} - \frac{1}{T_0}\right)}$

   $R_{thermistor}(100°C) = 100k\Omega \times e^{3977K \times \left(\frac{1}{373.15K} - \frac{1}{298.15K}\right)}$

   $R_{thermistor}(100°C) = 6.85 \ k\Omega$

5. Solve for $V_{thermistor}$ at T_{sp}.

   $V_{thermistor}(T_{sp}) = V_{cc} \times \frac{R_{thermistor}(T_{sp})}{R_1 + R_{thermistor}(T_{sp})}$

   $V_{thermistor}(100°C) = 5V \times \frac{6.85k\Omega}{10k\Omega + 6.85k\Omega} = 2.03V$

6. Solve for R_3 with the threshold voltage, V_{TH}, equal to $V_{thermistor}$. This ensures that $V_{thermistor}$ will always be larger than V_{TH} until the temperature switching point is exceeded.

   $R_3 = \frac{R_2 \times V_{TH}}{V_{cc} - V_{TH}}$

   $R_3 = \frac{1M\Omega \times 2.03V}{5V - 2.03V} = 685k\Omega$

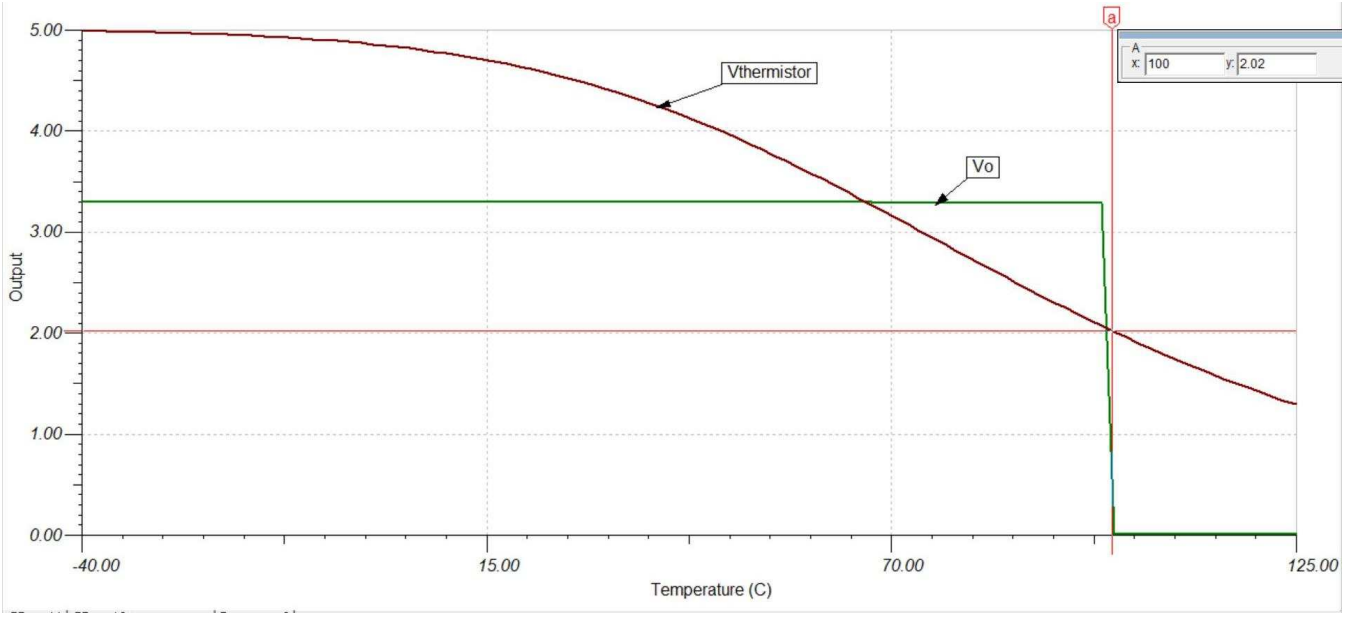
   $R_3 = 680k\Omega$ (Standard Value)

7. Select an appropriate pull up resistor, R_4. Here, V_{pu} = 3.3V (digital high for a microcontroller).

   $R_4 = 51k\Omega$ (Standard Value)

Design Simulations

DC Temperature Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See Circuit SPICE Simulation File SLVMCS1, www.ti.com/lit/zip/slvmcs1.

Design Featured Comparator

TLV7041	
Output Type	Open-Drain
V_{cc}	1.6V to 6.5V
V_{inCM}	Rail-to-rail
V_{os}	±100µV
V_{HYS}	7mV
I_q	335nA/Ch
t_{pd}	3µs
#Channels	1
www.ti.com/product/tlv7041	

Design Alternate Comparator

TLV1701	
Output Type	Open-Collector
V_{cc}	2.2V to 36V
V_{inCM}	Rail-to-rail
V_{os}	±500µV
V_{HYS}	N/A
I_q	55µA/Ch
t_{pd}	560ns
#Channels	1, 2, 4
	www.ti.com/product/tlv1701
	www.ti.com/product/tlv1701-q1

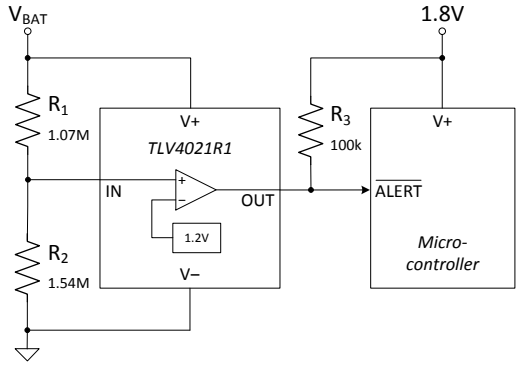
Undervoltage protection with comparator circuit

Design Goals

Battery Voltage Levels (V_{BAT})		Comparator Output Status (OUT)	
Undervoltage (V_{LOW})	Start-Up Operating Voltage (V_{HIGH})	Low Battery	Normal Operation
< 2.000V	> 2.034V	V_{OL} < 0.4V	V_{OH} = V_{PU} = 1.8V

Design Description

This undervoltage, protection circuit uses one comparator with a precision, integrated reference to create an alert signal at the comparator output (OUT) if the battery voltage sags below 2.0 V. The undervoltage alert in this implementation is ACTIVE LOW. So when the battery voltage drops below 2.0 V, the comparator output goes low, providing as an alert signal to whatever device is monitoring the output. Hysteresis is integrated in the comparator such that the comparator output will return to a logic high state when the battery voltage rises above 2.034 V. This circuit utilizes an open-drain output comparator in order to level shift the output high logic level for controlling a digital logic input pin. For applications needing to drive the gate of a MOSFET switch, a comparator with a push-pull output is preferred.



Design Notes

1. Select a comparator with a precision, integrated reference.
2. Select a comparator with an open-drain output stage for level-shifting.
3. Select values for the resistor divider so the critical undervoltage level occurs when the input to the comparator (IN) reaches the comparator's negative-going input threshold voltage (V_{IT-}).

Design Steps

1. Calculate the resistor divider ratio needed so the input to the comparator crosses V_{IT-} when V_{BAT} sags to the target undervoltage level (V_{LOW}) of 2.0V. V_{IT-} from the TLV4021R1 data sheet is 1.18V.

   $$V_{IT-} = \frac{R_2}{(R_1+R_2)} \times V_{LOW}$$

   $$\frac{R_2}{(R_1+R_2)} = \frac{V_{IT-}}{V_{LOW}} = \frac{1.18\ V}{2.00\ V} = 0.59$$

2. Confirm that the value of V_{LOW}, the voltage level where the undervoltage alert signal is asserted, is 2.0 V.

   $$V_{LOW} = \frac{R_1+R_2}{R_2} \times V_{IT-} = \frac{1}{0.59} \times 1.18\ \ V = 2.0\ \ V$$

3. Select values for R_1 and R_2 that yield the resistor divider ratio of 0.59 by using the following equation or using the online tool "Voltage Divider Calculator" at http://www.ti.com/download/kbase/volt/volt_div3.htm.

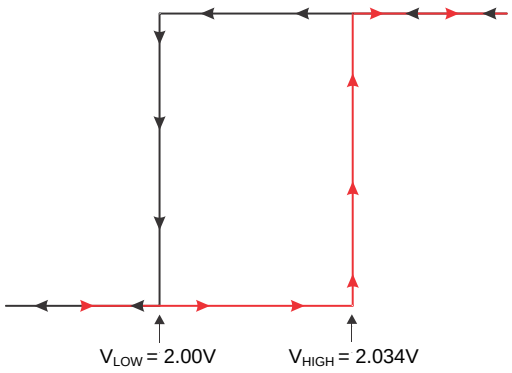
   If using the following equation, choose a value for R_2 in the Mega-ohm range and calculate for R1. In this example, a value of 1.54 M was chosen for R_2.

   $$R_1 = R_2\left(\frac{V_{LOW}}{V_{IT-}} - 1\right) = \ \ 1.54\ \ M\Omega\left(\frac{2\ V}{1.18\ V} - 1\right) = 1.07\ \ M\Omega$$

4. Verify that the current through the resistor divider is at least 100 times higher than the input bias current of the comparator. The resistors can have high values to minimize power consumption in the circuit without adding significant error to the resistor divider.

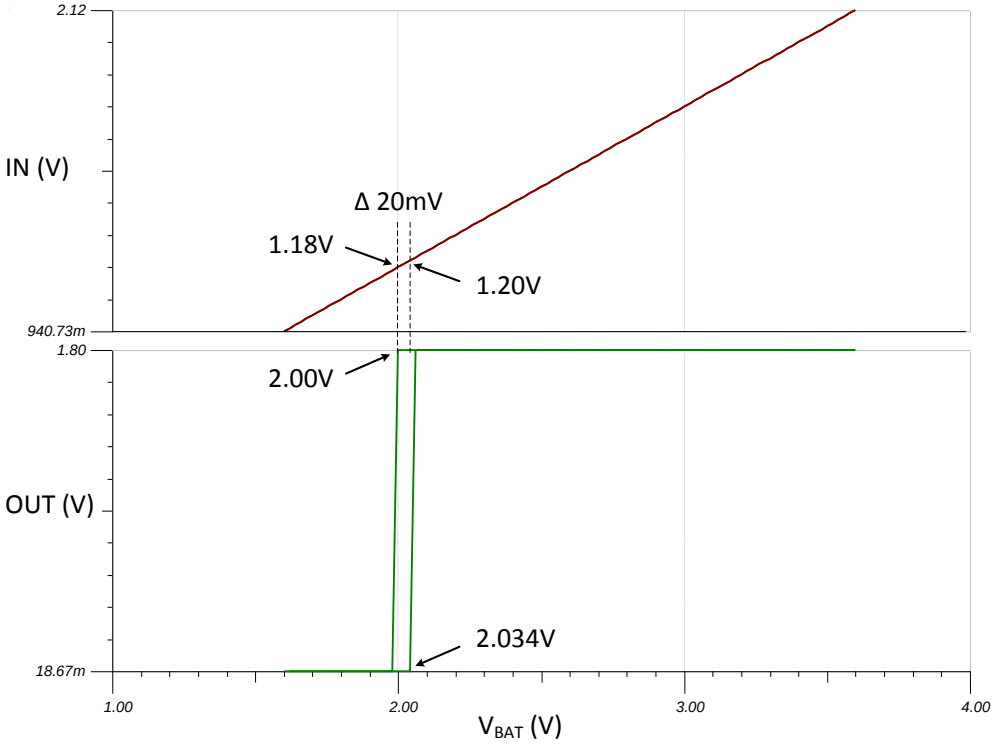
5. Calculate V_{HIGH}, the battery voltage where the undervoltage alert signal is de-asserted (returns to a logic high value). When the battery voltage reduces below the 2.0-V level or is ramping up at initial start-up, the comparator input needs to exceed (V_{IT+}), the positive-going input threshold voltage for the output to return to a logic high. V_{IT+} from the TLV4021R1 data sheet is 1.20V.

   $$V_{HIGH} = \frac{R_1+R_2}{R_2} \times V_{IT+} = \frac{1.07\ M\Omega+1.54\ M\Omega}{1.54\ M\Omega} \times 1.20V = 2.034\ \ V$$

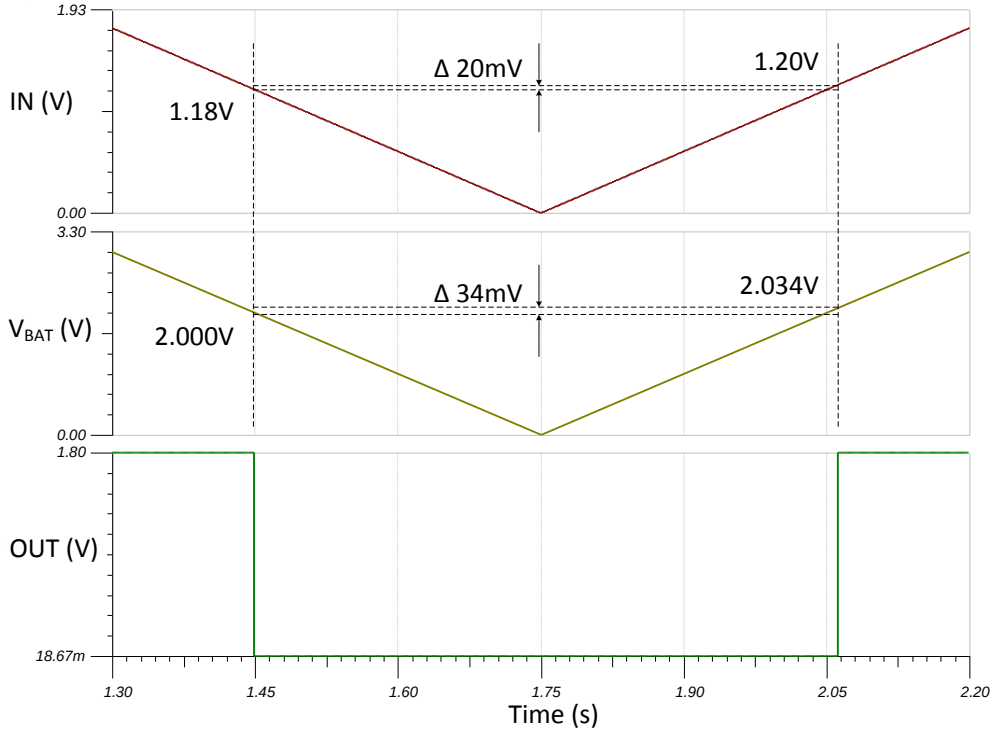


Design Simulations

DC Simulation Results



Transient Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File – SNOAA18
3. TI Precision Labs

Design Featured Comparator

TLV4021R1	
V_S	1.6V to 5.5V
V_{inCM}	Rail-to-rail
V_{OUT}	Open Drain
Integrated Reference	1.2V ±1% over temperature
Hysteresis	20 mV
I_Q	2.5 µA
$t_{PD(HL)}$	450 ns
www.ti.com/product/tlv4021	

Design Alternate Comparator

	TLV4041R1	TLV3011
V_S	1.6V to 5.5V	1.8V to 5.5V
V_{inCM}	Rail-to-rail	Rail-to-rail
V_{OUT}	Push-Pull	Open Drain
Integrated Reference	1.2 V ±1% over temperature	1.242 ±1% room temperature
Hysteresis	20mV	NA
I_Q	2.5µA	2.8µA
$t_{PD(HL)}$	450ns	6µs
	www.ti.com/product/tlv4041	www.ti.com/product/tlv3011

Window comparator circuit

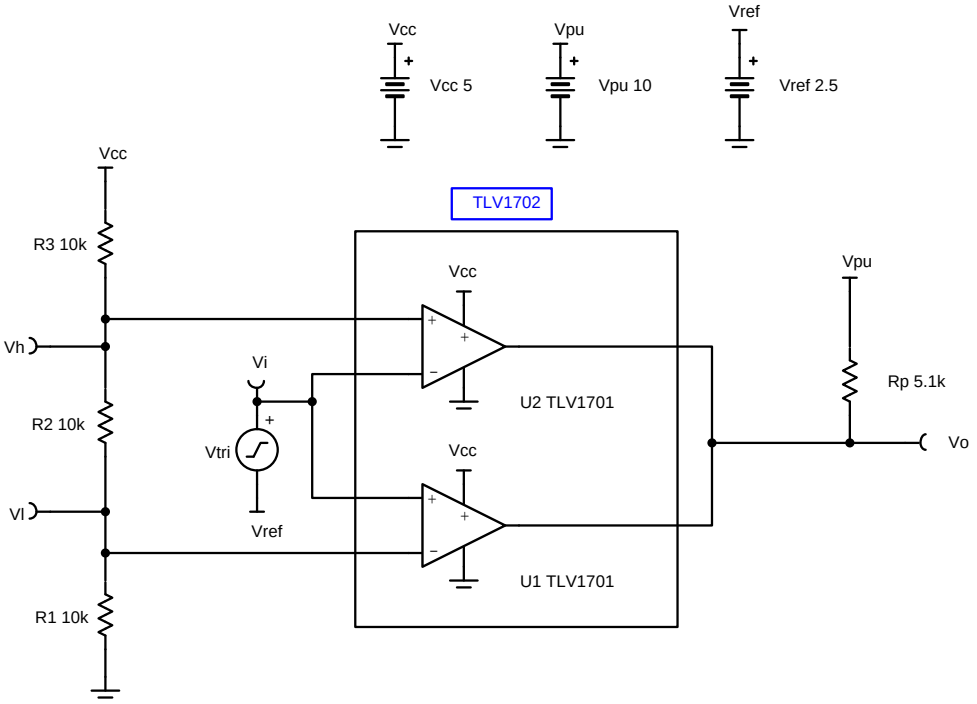
Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
0V	5V	0V	36V	5V	0V	2.5V

V_L (Lower Threshold)	V_H (Upper Threshold)	Upper to Lower Threshold Ratio
1.66V	3.33V	2

Design Description

This circuit utilizes two comparators in parallel to determine if a signal is between two reference voltages. If the signal is within the window, the output is high. If the signal level is outside of the window, the output is low. For this design, the reference voltages are generated from a single supply with voltage dividers.



Design Notes

1. The input should not exceed the common mode limitations of the comparators.
2. If higher pullup voltages are used, R_p should be sized accordingly to prevent large current draw. The TLV1701 supports pullup voltages up to 36V.
3. Comparator must be open-drain or open-collector to allow for the ORed output.

Design Steps

1. Define the upper (V_H) and lower (V_L) window voltages.

$$V_H = V_{cc} \times \frac{R_1 + R_2}{R_1 + R_2 + R_3} = 3.33\ V$$

$$V_L = V_{cc} \times \frac{R_1}{R_1 + R_2 + R_3} = 1.66\ V$$

$$\frac{V_H}{V_L} = 1 + \frac{R_2}{R_1} = \frac{3.33V}{1.66V} = 2$$

2. Choose resistor values to achieve the desired window voltages.

$$\frac{V_H}{V_L} = 1 + \frac{R_2}{R_1} = 2, \text{ so } R_2 = R_1$$

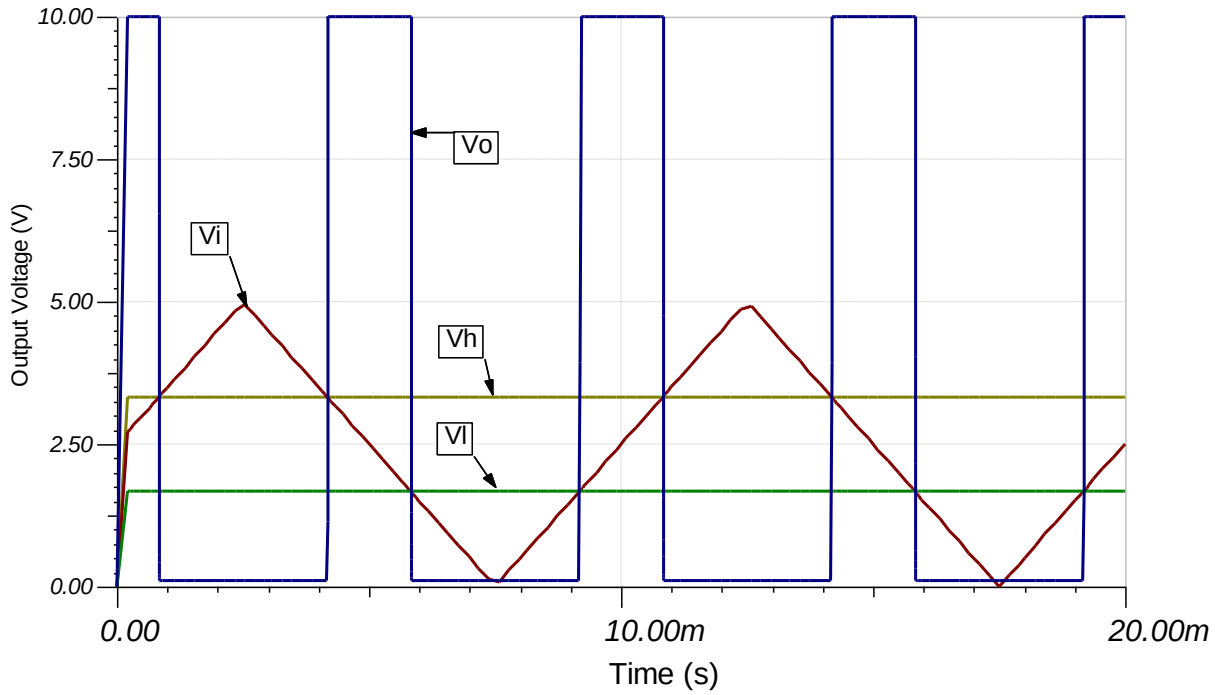
$$R_1 = R_2 = 10k\Omega \text{ (Selected standard values)}$$

$$R_3 = \frac{R_1 \times V_{cc}}{V_L} - (R_1 + R_2)$$

$$R_3 = \frac{10k\Omega \times 5V}{1.66V} - 20k\Omega = 10.12\ k\Omega \approx 10k\Omega \text{ (Standard Value)}$$

Design Simulations

Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC516.

See TIPD178, www.ti.com/tool/tipd178.

Design Featured Op Amp

TLV1702	
V_{cc}	2.2V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Open Collector (36V Max)
V_{os}	2.5mV
I_q	75µA/Ch
I_b	15nA
Rise Time	365ns
Fall Time	240ns
#Channels	1, 2, 4
www.ti.com/product/tlv1702	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

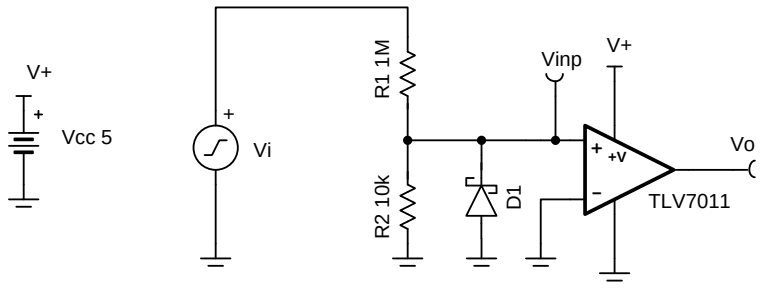
Zero crossing detection using comparator circuit

Design Goals

Supply			Input Signal		MAX AC Mains Leakage Current
V_{cc}	V_{ee}	Type	V_i	f	I_{ac}
5V	0V	Single	240V AC RMS	50Hz	<500µA

Design Description

The zero crossing detector circuit changes the comparator's output state when the AC input crosses the zero reference voltage. This is done by setting the comparator inverting input to the zero reference voltage and applying the attenuated input to the noninverting input.The voltage divider R_1 and R_2 attenuates the input AC signal. The diode D_1 is used to insure the noninverting input never goes below the negative input common mode limit of the comparator. Zero crossing detection is often used in power control circuits.



Design Notes

1. Some hysteresis should be used to prevent unwanted transitions due to the slow speed of the input signal.
2. Select a comparator with a large input common mode range
3. The phase inversion protection feature of the TLV7011 can prevent phase reversal in situations where the input goes outside of the input common mode limits
4. A diode should be used to protect the comparator when the input goes below the negative input common mode limit.

Design Steps

1. Calculate the peak value of the input signal.

   $V_p = V_{RMS} \text{ X } \sqrt{2} = 340V$

2. Select the resistor divider to attenuate the input 340V signal down to 3.4V in order to be within the positve common range of the comparator.

   $340V \text{ x } G = 3.4V$

   $G = 0.01 \frac{V}{V}$

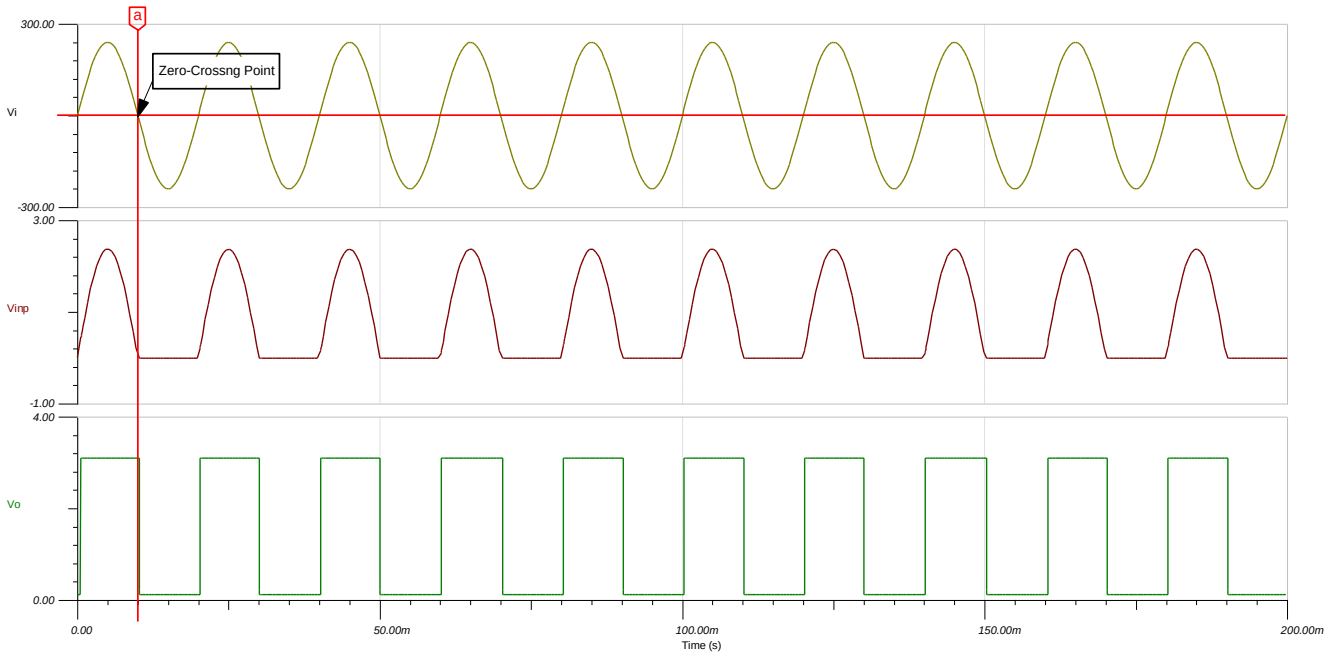
   $(\frac{R_2}{R_1+R_2}) = 0.01$

3. Select R_1 as 1MΩ and R_2 as 10kΩ (the closest 1% value).
4. Select the diode, D_1, in order to limit the negative voltage at the noninverting input. A zener diode with a voltage rating of 0.3V can be used.
5. Calculate the AC mains leakage current to check if it meets the leakage current design goal of less than 500µA.

   $I_{ac} = \frac{V_p}{R_1} = 340\mu A$

Design Simulations

Transient Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See circuit spice simulation file, SBOMAP5.

For more information on many comparator topics including hysteresis, propagation delay and input common mode range please see, TI Precision Labs.

Design Featured Comparator

TLV7011	
V_{ss}	1.6 to 5.5V
V_{inCM}	Rail-to-rail
t_{pd}	260ns
V_{os}	0.5mV
V_{HYS}	4mV
I_q	5µA
Output Type	Push-Pull
#Channels	1
www.ti.com/product/tlv7011	

Design Alternate Comparator

TLV3201	
V_{ss}	2.7 to 5.5V
V_{inCM}	Rail-to-rail
t_{pd}	40ns
V_{os}	1V
V_{HYS}	1.2mV
I_q	40µA
Output Type	Push-Pull
#Channels	1
www.ti.com/product/tlv3201	

Single-supply strain gauge bridge amplifier circuit

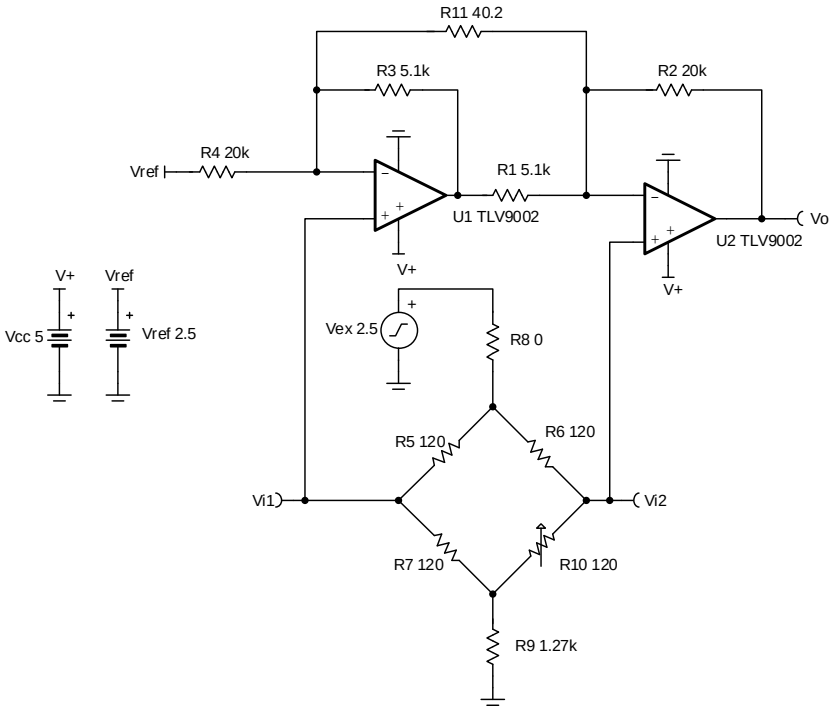
Design Goals

Input $V_{iDiff}(V_{i2} - V_{i1})$		Output		Supply		
V_{iDiff_Min}	V_{iDiff_Max}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
–2.22mV	2.27mV	225mV	4.72V	5V	0V	2.5V

Strain Gauge Resistance Variation (R_{10})	V_{cm}	Gain
115Ω – 125Ω	2.15V	1001V/V

Design Description

A strain gauge is a sensor whose resistance varies with applied force. The change in resistance is directly proportional to how much strain the sensor is experiencing due to the force applied. To measure the variation in resistance, the strain gauge is placed in a bridge configuration. This design uses a 2 op amp instrumentation circuit to amplify a differential signal created by the change in resistance of a strain gauge. By varying R_{10}, a small differential voltage is created at the output of the Wheatstone bridge which is fed to the 2 op amp instrumentation amplifier input. Linear operation of an instrumentation amplifier depends upon the linear operation of its primary building block: op amps. An op amp operates linearly when the input and output signals are within the device's input common–mode and output-swing ranges, respectively. The supply voltages used to power the op amps define these ranges.



Design Notes:

1. Resistors R_5, R_6 and R_7 of the Wheatstone bridge must match the stain gauge nominal resistance and must be equal to avoid creating a bridge offset voltage.
2. Low tolerance resistors must be used to minimize the offset and gain errors due to the bridge resistors.
3. V_{ex} sets the excitation voltage of the bridge and the common–mode voltage V_{cm}.
4. V_{ref} biases the output voltage of the instrumentation amplifier to mid–supply to allow differential measurements in the positive and negative directions.
5. R_{11} sets the gain of the instrumentation amplifier circuit.
6. R_8 and R_9 set the common–mode voltage of the instrumentation amplifier and limits the current through the bridge. This current determines the differential signal produced by the bridge. However, there are limitations on the current through the bridge due to self–heating effects of the bridge resistors and strain gauge.
7. Ensure that $R_1 = R_3$ and $R_2 = R_4$ and that ratios of R_2/R_1 and R_4/R_3 are matched to set the V_{ref} gain to 1V/V and maintain high DC CMRR of the instrumentation amplifier.
8. Linear operation is contingent upon the input common–mode and the output swing ranges of the op amps used. The linear output swing ranges are specified under the A_{ol} test conditions in the op amps datasheets.
9. Using high–value resistors can degrade the phase margin of the circuit and introduce additional noise in the circuit.

Design Steps:

1. Select R_5, R_6 and R_7 to match the stain gauge nominal resistance

   $$R_{gauge} = R_5 = R_6 = R_7 = 120\Omega$$

2. Choose R_9 to set the common mode voltage of the instrumentation amplifier at 2.15V

   $$V_{cm} = \frac{\frac{R_{bridge}}{2} + R_5}{R_{bridge} + R_8 + R_9} \times V_{ex}$$

   Where R_{bridge} = total resistance of the bridge

   Choose R_8 = $0\,\Omega$ to allow maximum current through the bridge

   $$V_{cm} = \frac{\frac{120\Omega \times 4}{2} + R_9}{120\Omega \times 4 + 0\Omega + R_9} \times 2.5\,V = 2.15\,V$$

   $$\frac{240 + R_9}{480 + 0\Omega + R_9} = \frac{2.15V}{2.5V} = 0.86$$

   $$0.14\,R_9 = 172.8 \rightarrow R_9 = \frac{172.8}{0.14} = 1.23\,k\Omega \rightarrow R_9 = 1.27\,k\,\Omega \text{ (Standard value)}$$

3. Calculate the gain required to produce the desired output voltage swing

   $$G = \frac{V_{oMax} - V_{oMin}}{V_{iDiff\_Min} - V_{iDiff\_Min}} = \frac{4.72V - 0.225V}{0.00222V - (-0.00227V)} = 1001\frac{V}{V}$$

4. Select R_1, R_2, R_3 and R_4. To set the V_{ref} gain at 1V/V and avoid degrading the instrumentation amplifier's CMRR, R_1 must equal R_3 and R_2 equal R_4.

   Choose $R_1 = R_3 = 5.1k\,\Omega$ and $R_3 = R_4 = 20\,k\,\Omega$ (Standard value)

5. Calculate R_{11} to meet the required gain

   $$G = 1 + \frac{R_4}{R_3} + \frac{2 \times R_2}{R_{11}} = 1001\frac{V}{V}$$

   $$G = 1 + \frac{20k\Omega}{5.1k\Omega} + \frac{2 \times R_2}{R_{11}} = 1001\frac{V}{V} \rightarrow 4.92 + \frac{40k\Omega}{R_{11}} = 1001\frac{V}{V} \rightarrow \frac{40k\Omega}{R_{11}} = 996.1 \rightarrow R_{11} = \frac{40k\Omega}{996.1} = 40.15\,\Omega \rightarrow R_{11} = 40.2\Omega \text{ (Standard Value)}$$

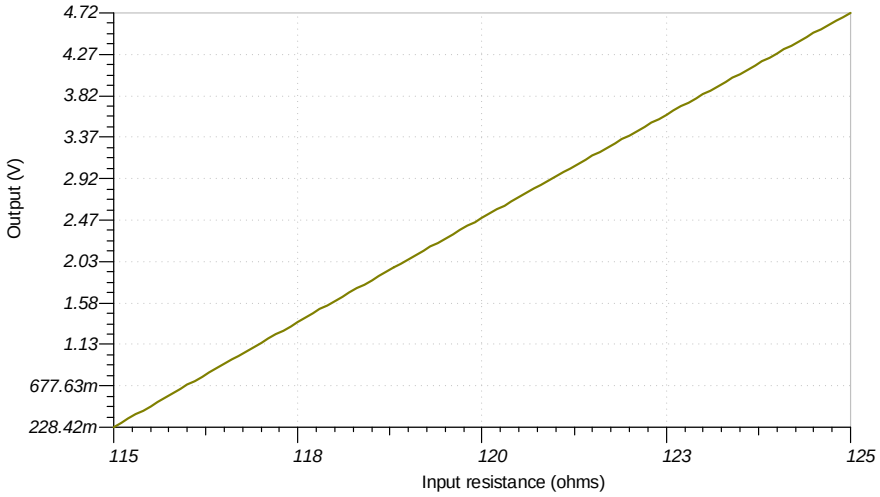
6. Calculate the current through the bridge

   $$I_{bridge} = \frac{V_{ex}}{R_8 + R_9 + R_{bridge}} = \frac{2.5V}{0\Omega + 1.27k\Omega + 120\Omega \times 4}$$

   $$I_{bridge} = \frac{2.5V}{1.27k\Omega + 480\Omega} \rightarrow I_{bridge} = 1.42mA$$

Design Simulations:

DC Simulation Results



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOMAU4
3. TI Precision Designs TIPD170
4. TI Precision Labs
5. V_{CM} vs. V_{OUT} plots for instrumentation amplifiers with two op amps

Design Featured Op Amp:

TLV9002	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–Rail
V_{os}	0.4mV
I_q	0.06mA
I_b	5pA
UGBW	1MHz
SR	2V/µs
#Channels	1,2,4
www.ti.com/product/tlv9002	

Design Alternate Op Amp:

OPA376	
V_{ss}	2.2V to 5.5V
V_{inCM}	(V_{ee}–0.1V) to (V_{cc}–1.3V)
V_{out}	Rail–to–Rail
V_{os}	0.005mV
I_q	0.76mA
I_b	0.2pA
UGBW	5.5MHz
SR	2V/µs
#Channels	1,2,4
www.ti.com/product/opa376	

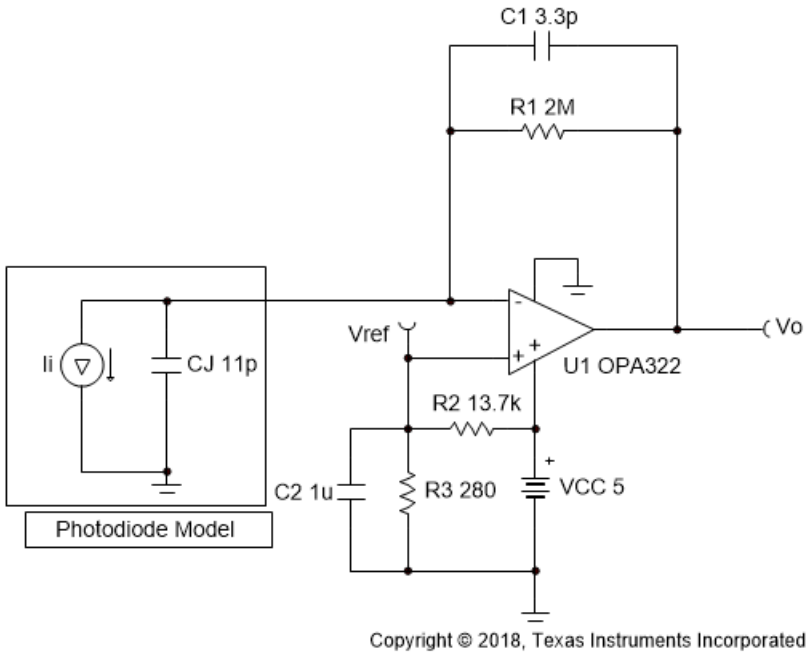
Photodiode amplifier circuit

Design Goals

Input		Output		BW	Supply		
I_{iMin}	I_{iMax}	V_{oMin}	V_{oMax}	f_p	V_{cc}	V_{ee}	V_{ref}
0A	2.4µA	100mV	4.9V	20kHz	5V	0V	0.1V

Design Description

This circuit consists of an op amp configured as a transimpedance amplifier for amplifying the light-dependent current of a photodiode.



Design Notes

1. A bias voltage (V_{ref}) prevents the output from saturating at the negative power supply rail when the input current is 0A.
2. Use a JFET or CMOS input op amp with low bias current to reduce DC errors.
3. Set output range based on linear output swing (see A_{ol} specification).

Design Steps

1. Select the gain resistor.

$$R_1 = \frac{V_{oMax} - V_{oMin}}{I_{iMax}} = \frac{4.9V - 0.1V}{2.4\mu A} = 2M\Omega$$

2. Select the feedback capacitor to meet the circuit bandwidth.

$$C_1 \leq \frac{1}{2 \times \pi \times R_1 \times f_p}$$

$$C_1 \leq \frac{1}{2 \times \pi \times 2M\Omega \times 20kHz} \leq 3.97pF \approx 3.3pF \text{ (Standard Value)}$$

3. Calculate the necessary op amp gain bandwidth (GBW) for the circuit to be stable.

$$GBW > \frac{C_i + C_1}{2 \times \pi \times R_1 \times C_1^2} > \frac{20pF + 3.3pF}{2 \times \pi \times 2M\Omega \times (3.3pF)^2} > 170kHz$$

where $C_i = C_j + C_d + C_{cm} = 11pF + 5pF + 4pF = 20pF$ given

- C_j: Junction capacitance of photodiode
- C_d: Differential input capacitance of the amplifier
- C_{cm}: Common-mode input capacitance of the inverting input

4. Calculate the bias network for a 0.1-V bias voltage.

$$R_2 = \frac{V_{cc} - V_{ref}}{V_{ref}} \times R_3$$

$$R_2 = \frac{5V - 0.1V}{0.1V} \times R_3$$

$$R_2 = 49 \times R_3$$

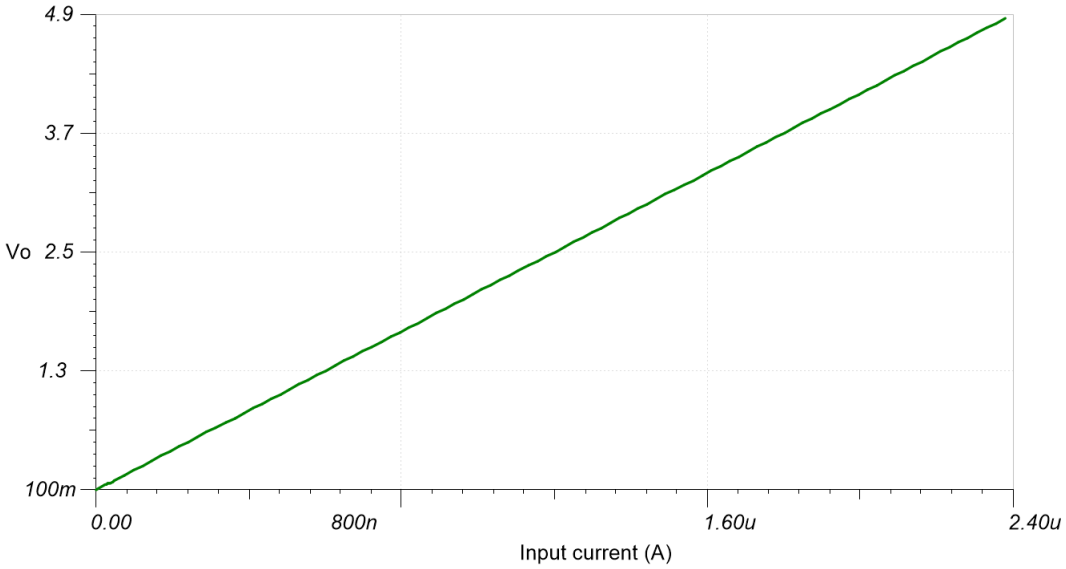
Closest 1% resistor values that yield this relationship are
$R_2 = 13.7k\Omega$ and $R_3 = 280\Omega$

5. Select C_2 to be 1µF to filter the V_{ref} voltage. The resulting cutoff frequency is:

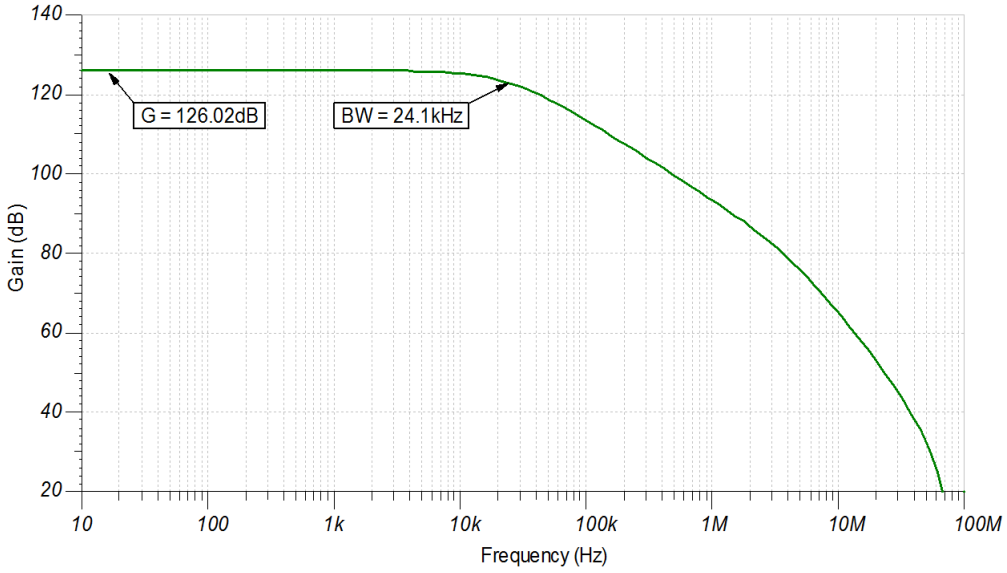
$$f_p = \frac{1}{2 \times \pi \times C_2 \times (R_2 \| R_3)} = \frac{1}{2 \times \pi \times 1\ \mu F \times (13.7k \| 280)} = 580Hz$$

Design Simulations

DC Simulation Results



AC Simulation Results



Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC517.

See TIPD176, www.ti.com/tool/tipd176.

Design Featured Op Amp

OPA322	
V_{cc}	1.8V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	0.5mV
I_q	1.6mA/Ch
I_b	0.2pA
UGBW	20MHz
SR	10V/µs
#Channels	1, 2, 4
www.ti.com/product/opa322	

Design Alternate Op Amp

LMP7721	
V_{cc}	1.8V to 5.5V
V_{inCM}	V_{ee} to (V_{cc} –1V)
V_{out}	Rail–to–rail
V_{os}	26µV
I_q	1.3mA/Ch
I_b	3fA
UGBW	17MHz
SR	10.43V/µs
#Channels	1
www.ti.com/product/lmp7721	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.

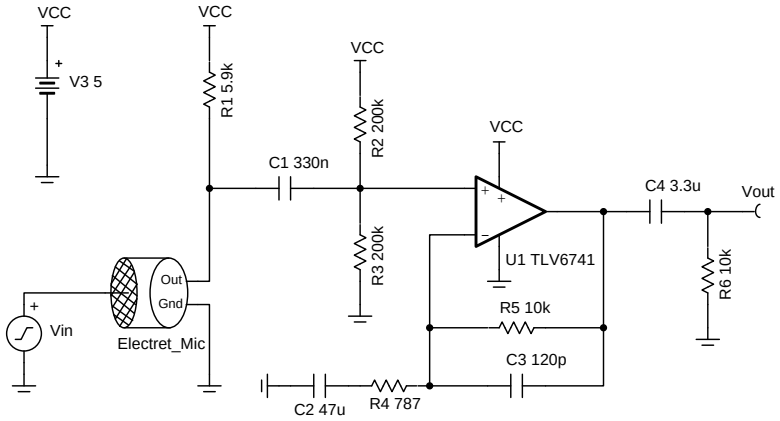
Non-inverting microphone pre-amplifier circuit

Design Goals

Input Pressure (Max)	Output Voltage (Max)	Supply		Frequency Response Deviation	
100dB SPL (2 Pa)	1.228V_{rms}	V_{cc}	V_{ee}	@20Hz	@20kHz
		5V	0V	–0.5dB	–0.1dB

Design Description

This circuit uses a non–inverting amplifier circuit configuration to amplify the microphone output signal. This circuit has very good magnitude flatness and exhibits minor frequency response deviations over the audio frequency range. The circuit is designed to be operated from a single 5V supply.



Design Notes

1. Operate within the op amp linear output operating range, which is usually specified under the A_{OL} test conditions.
2. Use low–K capacitors (tantalum, C0G, and so forth) and thin film resistors help to decrease distortion.
3. Use a battery to power this circuit to eliminate distortion caused by switching power supplies.
4. Use low value resistors and low noise op amps for low noise designs.
5. The common mode voltage is equal to the DC bias voltage set using the resistor divider plus any variation caused by the microphone output voltage. For op amps with a complementary pair input stage it is recommended to keep the common mode voltage away from the cross over region to eliminate the possibility of cross over distortion.
6. Resistor R_1 is used to bias the microphone internal JFET transistor to achieve the bias current specified by the microphone.
7. The equivalent input resistance is determined by R_1, R_2, R_3. Use large value resistors for R_2 and R_3 to increase the input resistance.
8. The voltage connected to R_1 to bias the microphone does not have to be the same as the op amp supply voltage. Using a higher voltage supply for the microphone bias allows for a lower bias resistor value.

Design Steps

This design procedure uses the microphone specifications provided in the following table.

Microphone Parameter	Value
Sensitivity @ 94dB SPL (1 Pa)	–35 ± 4 dBV
Current Consumption (Max)	0.5mA
Impedance	2.2kΩ
Standard Operating Voltage	2Vdc

1. Convert the sensitivity to volts per Pascal.
$$10^{\frac{-35dB}{20}} = 17.78\frac{mV}{Pa}$$
2. Convert volts per Pascal to current per Pascal.
$$\frac{17.78\frac{mV}{Pa}}{2.2k\Omega} = 8.083\frac{\mu A}{Pa}$$
3. Max output current occurs at max pressure 2Pa.
$$I_{Max} = 2Pa \times 8.083\frac{\mu A}{Pa} = 16.166\mu A$$
4. Calculate bias resistor. In the following equation, Vmic is microphone standard operating voltage.
$$R_1 = \frac{V_{cc} - V_{mic}}{I_s} = \frac{5V-2V}{0.5mA} = 6k\Omega \approx 5.9k\Omega \text{ (Standard Value)}$$
5. Set the amplifier input common mode voltage to mid–supply voltage. The equivalent resistance of R_2 in parallel with R_3 should be 10 times larger than R1 so that a majority of the microphone current flows through R_1.
$$R_{eq} = R2 \| R3 > 10 \times R1 = 100k\Omega$$
Choose $R_2 = R_3 = 200k\Omega$
6. Calculate the maximum input voltage.
$$R_{in} = R1 \| R_{eq} = 5.9k\Omega \| 100k\Omega = 5.571k\Omega$$
$$V_{in} = I_{max} \times R_{in} = 16.166uA \times 5.571k\Omega = 90.067mV$$
7. Calculate gain required to produce the largest output voltage swing.
$$Gain = \frac{V_{outmax}}{V_{in}} = \frac{1.228V}{90.067mV} = 13.634\frac{V}{V}$$
8. Calculate R_4 to set the gain calculated in step 7. Select feedback resistor R_5 as 10kΩ.
$$R_4 = \frac{R_5}{Gain-1} = \frac{10k\Omega}{13.634-1} = 791\Omega \approx 787\Omega \text{ (Standard Values)}$$
The final gain of this circuit is:
$$Gain = 20\log\frac{Vout}{Vin} = 20\log\frac{16.166uA \times 5.571k\Omega \times \left(1+\frac{10k\Omega}{787\Omega}\right)}{2V} = -4.191dB$$
9. Calculate the corner frequency at low frequency according to the allowed deviation at 20 Hz. In the following equation, G_pole1 is the gain contributed by each pole at frequency “f”. Note that you divide by three because there are three poles.
$$f_c = f\sqrt{\left(\frac{1}{G\_pole1}\right)^2 - 1} = 20Hz\sqrt{\left(\frac{1}{10^{\frac{-0.5/3}{20}}}\right)^2 - 1} = 3.956Hz$$
10. Calculate C_1 based on the cut off frequency calculated in step 9.
$$C_1 = \frac{1}{2\pi \times Req \times f_c} = \frac{1}{2\pi \times 100k\Omega \times 3.956Hz} = 0.402\mu F \approx 0.33\mu F \text{ (Standard Value)}$$
11. Calculate C_2 based on the cut off frequency calculated in step 9.
$$C_2 = \frac{1}{2\pi \times R4 \times f_c} = \frac{1}{2\pi \times 787\Omega \times 3.956Hz} = 51.121\mu F \approx 47\mu F \text{ (Standard Value)}$$
12. Calculate the high frequency pole according to the allowed deviation at 20 kHz. In the following equation, G_pole2 is the gain contributed by each pole at frequency “f”.
$$f_p = \frac{f}{\sqrt{\left(\frac{1}{G\_pole2}\right)^2 - 1}} = \frac{20kHz}{\sqrt{\left(\frac{1}{10^{\frac{-0.1}{20}}}\right)^2 - 1}} = 131.044kHz$$

13. Calculate C3 to set the cut off frequency calculated in step 12.

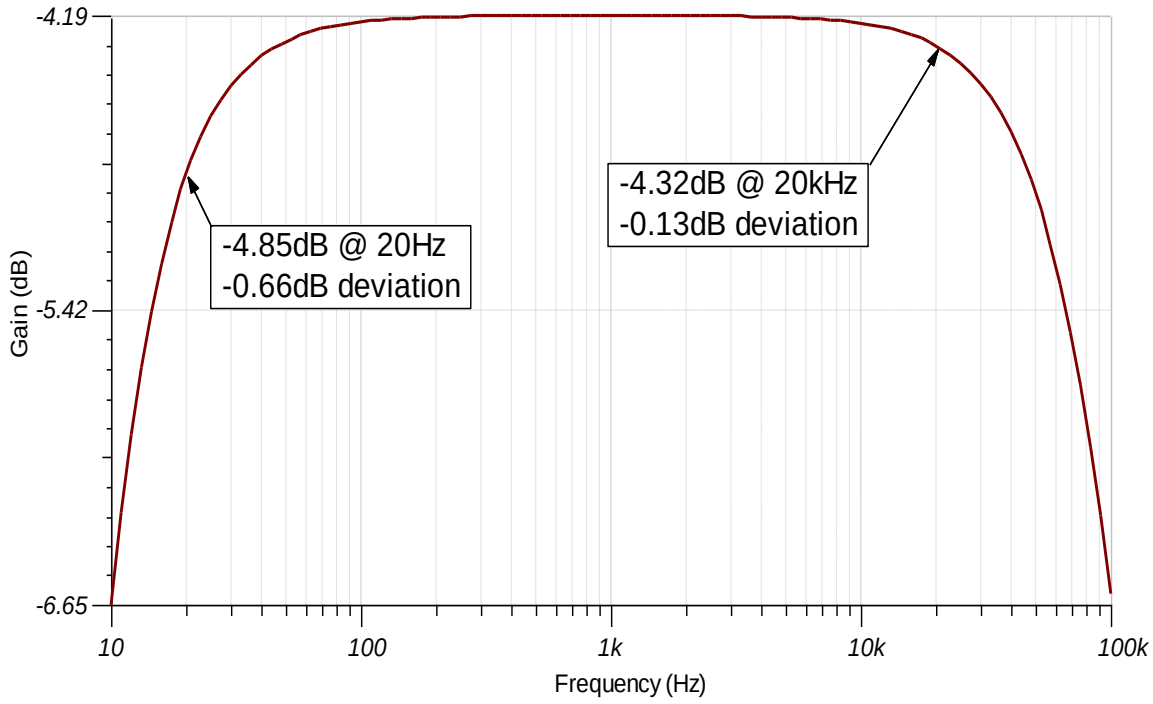
$$C_3 = \frac{1}{2\pi \times R_5 \times f_p} = \frac{1}{2\pi \times 10k\Omega \times 131.044kHz} = 121.451pF \approx 120pF \text{ (Standard Value)}$$

14. Calculate the output capacitor, C_4, based on the cut off frequency calculated in step 9. Assume the output load R_6 is 10kΩ.

$$C_4 = \frac{1}{2\pi \times R_6 \times f_c} = \frac{1}{2\pi 10k\Omega \times 3.956Hz} = 4.023\mu F \approx 3.3\mu F \text{ (Standard Value)}$$

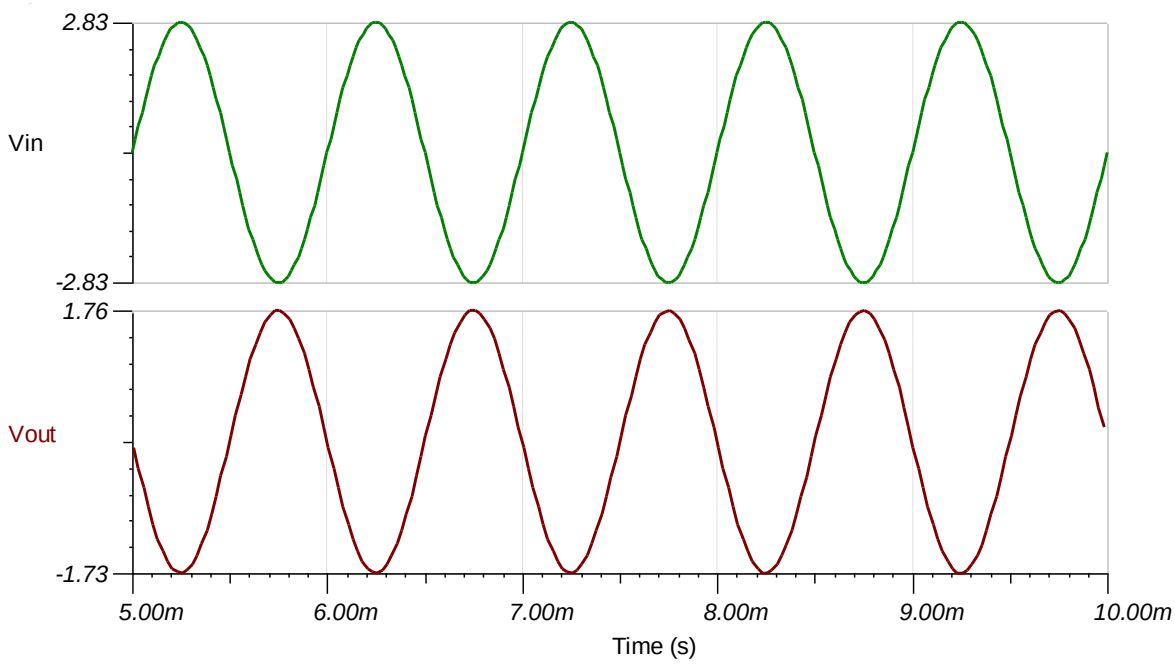
Design Simulations

AC Simulation Results



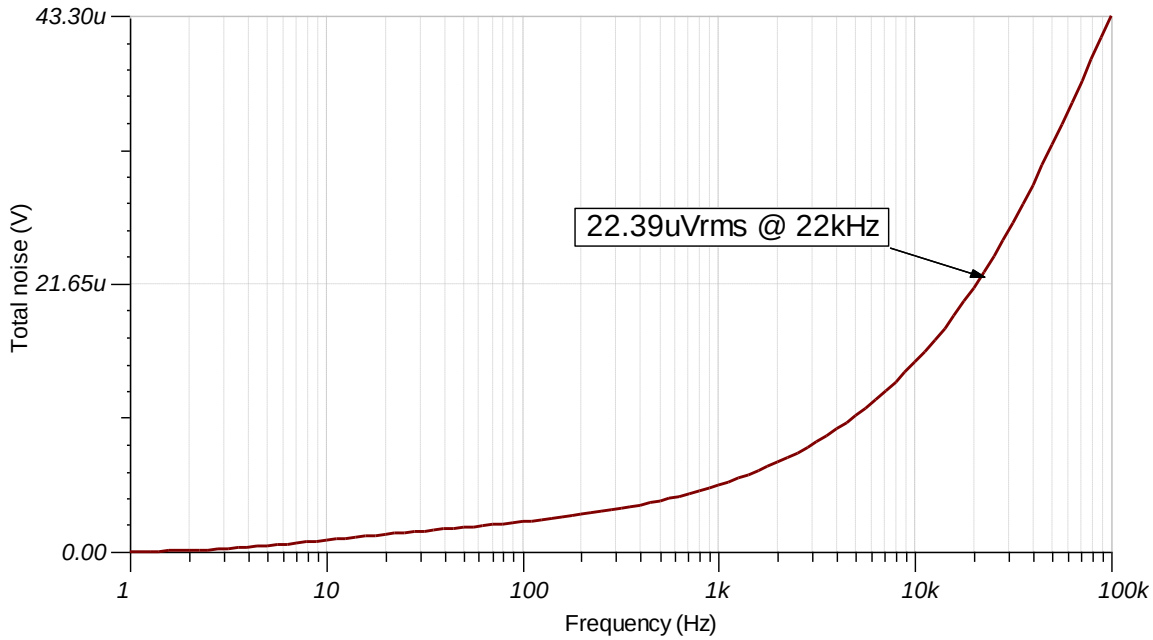
Transient Simulation Results

The input voltage represents the SPL of an input signal to the microphone. A 1 V_{rms} input signal represents 1 Pascal.



Noise Simulation Results

The following simulation results show 22.39uVrms of noise at 22kHz. The noise is measured at a bandwidth of 22kHz to represent the measured noise using an audio analyzer with the bandwidth set to 22kHz.



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOC525
3. TI Precision Designs TIPD181
4. TI Precision Labs

Design Featured Op Amp

TLV6741	
V_{ss}	1.8V to 5.5V
V_{inCM}	(Vee) to (Vcc –1.2V)
V_{out}	Rail–to–rail
V_{os}	150µV
I_q	890uA/Ch
I_b	10pA
UGBW	10MHz
SR	4.75V/µs
#Channels	1
www.ti.com/product/tlv6741	

Design Alternate Op Amp

OPA320	
V_{ss}	1.8V to 5.5V
V_{inCM}	Rail–to–rail
V_{out}	Rail–to–rail
V_{os}	40µV
I_q	1.5mA/Ch
I_b	0.2pA
UGBW	20MHz
SR	10V/µs
#Channels	1, 2
www.ti.com/product/opa320	

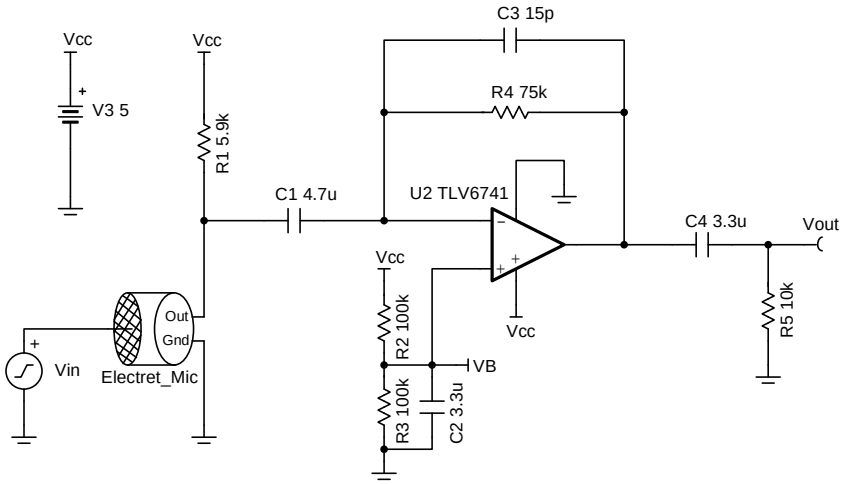
TIA microphone amplifier circuit

Design Goals

Input pressure (Max)	Output Voltage (Max)	Supply		Frequency Response Deviation	
100dB SPL(2Pa)	1.228V_{rms}	V_{cc}	V_{ee}	@ 20Hz	@20kHz
		5V	0V	–0.5dB	–0.1dB

Design Description

This circuit uses an op amp in a transimpedance amplifier configuration to convert the output current from an electret capsule microphone into an output voltage. The common mode voltage of this circuit is constant and set to mid–supply eliminating any input–stage cross over distortion.



Design Notes

1. Use the op amp in the linear output operating range, which is usually specified under the A_{OL} test conditions.

2. Use low–K capacitors (tantalum, C0G, etc.) and thin film resistors help to decrease distortion.

3. Use a battery to power this circuit to eliminate distortion caused by switching power supplies.

4. Use low value resistors and low noise op amp to achieve high performance low noise designs.

5. The voltage connected to R_1 to bias the microphone does not have to match the supply voltage of the op amp. Using a larger microphone bias voltage allows for a larger value or R_1 which decreases the noise gain of the op amp circuit while still maintaining normal operation of the microphone.

6. Capacitor C_1 should be large enough that its impedance is much less than resistor R_1 at audio frequency. Pay attention to the signal polarity when using tantalum capacitors.

Design Steps

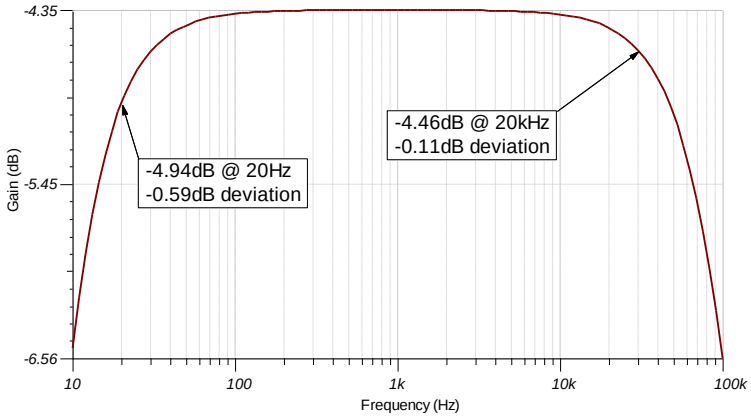
The following microphone is chosen as an example to design this circuit.

Microphone parameter	Value
Sensitivity @ 94dB SPL (1 Pa)	–35 ± 4 dBV
Current Consumption (Max)	0.5mA
Impedance	2.2kΩ
Standard Operating Voltage	$2V_{dc}$

1. Convert the sensitivity to volts per Pascal.
   $10^{\frac{-35dB}{20}} = 17.78\,mV / Pa$
2. Convert volts per Pascal to current per Pascal.
   $\frac{17.78mV / Pa}{2.2k\Omega} = 8.083\,\mu A / Pa$
3. Max output current occurs at max sound pressure level of 2Pa.
   $I_{Max} = 2Pa \times 8.083\,\mu A / Pa = 16.166\,\mu A$
4. Calculate the value of resistor R_4 to set the gain
   $R_4 = \frac{V_{max}}{I_{max}} = \frac{1.228V}{16.166\mu A} = 75.961\,k\Omega \approx 75k\Omega$ (Standard value)
   The final signal gain is:
   $Gain = 20 \times \log \frac{V_{out}}{V_{in}} = 20 \times \log \frac{16.166\mu A \times 75k\Omega}{2V} = -4.347\,dB$
5. Calculate the value for the bias resistor R_1. In the following equation, Vmic is the standard operating voltage of the microphone
   $R_1 = \frac{V_{cc} - V_{mic}}{I_s} = \frac{5V - 2V}{0.5mA} = 6k\Omega \approx 5.9\,k\Omega$ (Standard value)
6. Calculate the high frequency pole according to the allowed deviation at 20 kHz. In the following equation, G_pole1 is the gain at frequency "f".
   $f_p = \frac{f}{\sqrt{(\frac{1}{G\_pole1})^2 - 1}} = \frac{20kHz}{\sqrt{(\frac{1}{10^{\frac{-0.1}{20}}})^2 - 1}} = 131.044\,kHz$
7. Calculate C_3 based on the pole frequency calculated in step 6.
   $C_3 = \frac{1}{2\pi \times f_p \times R_4} = \frac{1}{2\pi \times 131.044kHz \times 75k\Omega} = 16.194\,pF \approx 15pF$ (Standard value)
8. Calculate the corner frequency at low frequency according to the allowed deviation at 20 Hz. In the following equation, G_pole2 is the gain contributed by each pole at frequency "f" respectively. There are two poles, so divided by two.
   $f_c = f \times \sqrt{(\frac{1}{G\_pole2})^2 - 1} = 20Hz \times \sqrt{(\frac{1}{10^{\frac{-0.5/2}{20}}})^2 - 1} = 4.868\,Hz$
9. Calculate the input capacitor C_1 based on the cut off frequency calculated in step 8.
   $C_1 = \frac{1}{2\pi \times R_1 \times f_c} = \frac{1}{2\pi \times 5.9k\Omega \times 4.868Hz} = 5.541\,\mu F \approx 4.7\,\mu F$ (Standard value)
10. Assuming the output load R_5 is 10kΩ, calculate the output capacitor C_4 based on the cut off frequency calculated in step 8.
    $C_4 = \frac{1}{2\pi \times R_5 \times f_c} = \frac{1}{2\pi \times 10k\Omega \times 4.868Hz} = 3.269\,\mu F \approx 3.3\,\mu F$ (Standard value)
11. Set the amplifier input common mode voltage to mid–supply voltage. Select R_2 and R_3 as 100kΩ. The equivalent resistance equals to the parallel combination of the two resistors:
    $R_{eq} = R_2 || R_3 = 100k\Omega || 100k\Omega = 50k\Omega$
12. Calculate the capacitor C_2 to filter the power supply and resistor noise. Set the cutoff frequency to 1Hz.
    $C_2 = \frac{1}{2\pi \times (R_2 || R_3) \times 1Hz} = \frac{1}{2\pi \times (100k\Omega || 100k\Omega) \times 1Hz} = 3.183\,\mu F \approx 3.3\,\mu F$ (Standard value)

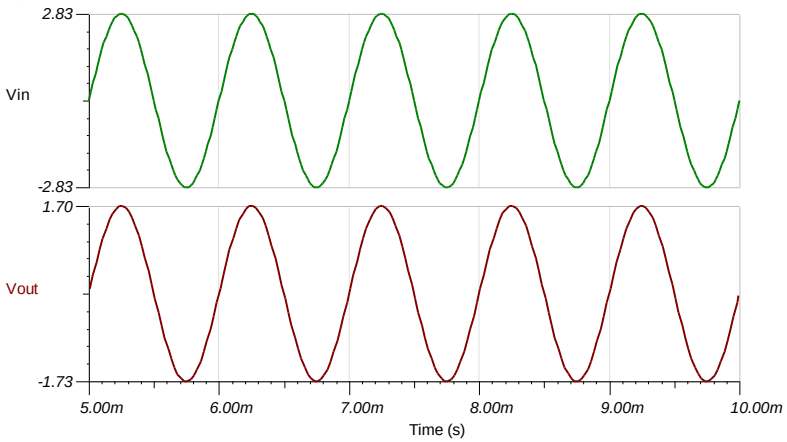
Design Simulations

AC Simulation Results



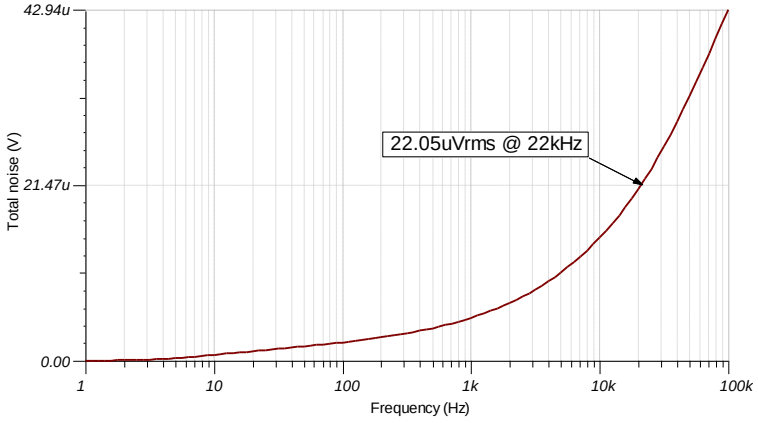
Transient Simulation Results

The input voltage represents the SPL of an input signal to the microphone. A 2 V_{rms} input signal represents 2 Pascal.



Noise Simulation Results

The following simulation results show 22.39µV_{rms} of noise at 22kHz. The noise is measured at a bandwidth of 22kHz to represent the measured noise using an audio analyzer with the bandwidth set to 22kHz.



References:

1. Analog Engineer's Circuit Cookbooks
2. SPICE Simulation File SBOC526
3. TI Precision Designs TIPD181
4. TI Precision Labs

Design Featured Op Amp

TLV6741	
V_{ss}	1.8V to 5.5V
V_{inCM}	V_{ee} to V_{cc}–1.2V
V_{out}	Rail–to–rail
V_{os}	150µV
I_q	890µA/Ch
I_b	10pA
UGBW	10MHz
SR	4.75V/µs
#Channels	1
www.ti.com/product/tlv6741	

Design Alternate Op Amp

	OPA172	OPA192
V_{ss}	4.5V to 36V	4.5V to 36V
V_{inCM}	V_{ee}–0.1V to V_{cc}–2V	V_{ee}–0.1V to V_{cc}+0.1V
V_{out}	Rail–to–rail	Rail–to–rail
V_{os}	±200µV	±5µV
I_q	1.6mA/Ch	1mA/Ch
I_b	8pA	5pA
UGBW	10MHz	10MHz
SR	10V/µs	20V/µs
#Channels	1, 2, 4	1, 2, 4
	www.ti.com/product/opa172	www.ti.com/product/opa192

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES “AS IS” AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI’s products are provided subject to TI’s Terms of Sale (www.ti.com/legal/termsofsale.html) or other applicable terms available either on ti.com or provided in conjunction with such TI products. TI’s provision of these resources does not expand or otherwise alter TI’s applicable warranties or warranty disclaimers for TI products.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265